DEVELOPMENT OF ULTRASONIC ORBITAL MICROFORMING (Vibration forming and modal analysis)

INTERMEDIATE INTEGRATING PROJECT

WARSAW UNIVERSITY OF TECHNOLOGY –

Faculty of production 2019/20

Student – Jorge Blasco Gracia (K-5513)

Teacher – dr hab. inz. Wojciech Presz

INDEX

THEORETICAL ANALYSIS OF THE PROCESS

DEFINITION OF THE PROBLEM

This work corresponds to the subject Development of Ultrasonic Orbital Microforming (subject of the Intermediate Integrating Project course), specifically the study of the formation of a specific shape in a small cylindrical piece due to a vibration movement induced in a sonotrode. Later on, it will be seen that this vibration movement is transformed due to frequencies close to synchronization in a movement that combines this vibration with a certain orbit on the upper surface of this small cylindrical piece. The study will be separated into two large sections which will be brought together at the end to obtain overall conclusions (although due to time constraints, it has not been possible to bring the simulations together for both purposes).

The two parts into which this phenomenon will be divided in order to study it in a more particular way are the Ultrasonic Vibration Analysis whose objective is to find the modal analysis of the sonotrode that will transmit the movement from the base to the piece to be modelled. This part will be developed in this report. On the other hand, to find the reason why the small cylindrical part ends up with the shape it does (this we know due to the practical analysis by means of camera and visual analysis in the laboratory) we are also going to carry out a separate study on the orbital process, (Project, 2020). Because this process studies the possible movement associated with the end of this sonotrode and its contact with the part, it will be carried out in 3D in its entirety. Therefore the two arts are:

Orbital process

On this part of the analysis, three-dimensional studies will be carried out on samples of small dimensions (mainly 1x1mm cylinders) whose main material will be aluminium, cooper and stainless steel, since this materials have proper mechanical properties for carrying out this type of manufacturing. This studies will mainly be focused on the 3D commented analysis in which deformations are applied to each part using orbital trajectories with the target of achieving simulations that allow the effects and movements that have previously been obtained experimentally by this type of producing processes. They will be treated in the most similar way to this experiment trying to achieve the movement that cause the same deformation than in this practical experiment.

Ultrasonic vibrations analysis

This part, developed throughout this work, will focus on finding the vibrations that cause this barrel-shaped phenomenon in the initial cylinder and mainly, find out how to obtain rotation. The cylinder to be deformed is the same as the one mentioned in the previous part (1x1 mm) and the object that transmits the movement from the base to the cylinder itself, also previously mentioned, is a sonotrode. A sonotrode in itself, is composed of 3 different parts, the main body, an object that usually adds the necessary weight to the instrument and finally the final part that carries the desired shape. Due to the short time I have, these three parts are going to be treated as a single piece of the same material, so the general shape will be preserved and will be studied in the last part of this report. For a more developed description of what is treated in this work, see the part **¡Error! No se encuentra el origen de la referencia.** where everything is widely developed.

The following is a short introduction to the theory concerning this work:

MODAL ANALYSIS

Modal analysis is the study of the dynamic properties of systems in the frequency domain. Examples would include measuring the vibration of a car's body when it is attached to a shaker, or the noise pattern in a room when excited by a loudspeaker.

Modern day experimental modal analysis systems are composed of 1) sensors such as transducers (typically accelerometers, load cells), or non-contact via a Laser vibrometer, or stereo photogrammetric cameras 2) data acquisition system and an analog-to-digital converter front end (to digitize analog instrumentation signals) and 3) host PC (personal computer) to view the data and analyse it.

Classically this was done with a SIMO (single-input, multiple-output) approach, that is, one excitation point, and then the response is measured at many other points. In the past a hammer survey, using a fixed accelerometer and a roving hammer as excitation, gave a MISO (multiple-input, single-output) analysis, which is mathematically identical to SIMO, due to the principle of reciprocity. In recent years MIMO (multi-input, multiple-output) have become more practical, where partial coherence analysis identifies which part of the response comes from which excitation source. Using multiple shakers leads to a uniform distribution of the energy over the entire structure and a better coherence in the measurement. A single shaker may not effectively excite all the modes of a structure.

Typical excitation signals can be classed as impulse, broadband, swept sine, chirp, and possibly others. Each has its own advantages and disadvantages.

The analysis of the signals typically relies on Fourier analysis. The resulting transfer function will show one or more resonances, whose characteristic mass, frequency and damping can be estimated from the measurements.

The animated display of the mode shape is very useful to NVH (noise, vibration, and harshness) engineers.

The results can also be used to correlate with finite element analysis normal mode solutions.

In structural engineering, modal analysis uses the overall mass and stiffness of a structure to find the various periods at which it will naturally resonate. These periods of vibration are very important to note in earthquake engineering, as it is imperative that a building's natural frequency does not match the frequency of expected earthquakes in the region in which the building is to be constructed. If a structure's natural frequency matches an earthquake's frequency, the structure may continue to resonate and experience structural damage. Modal analysis is also important in structures such as bridges where the engineer should attempt to keep the natural frequencies away from the frequencies of people walking on the bridge. This may not be possible and for this reasons when groups of people are to walk along a bridge, for example a group of soldiers, the recommendation is that they break their step to avoid possibly significant excitation frequencies. Other natural excitation frequencies may exist and may excite a bridge's natural modes. Engineers tend to learn from such examples (at least in the short term) and more modern suspension bridges take account of the potential influence of wind through the shape of the deck, which might be designed in aerodynamic terms to pull the deck down against the support of the structure rather than allow it to lift. Other aerodynamic loading issues are dealt with by minimizing the area of the structure projected to the oncoming wind and to reduce wind generated oscillations of, for example, the hangers in suspension bridges.

Although modal analysis is usually carried out by computers, it is possible to hand-calculate the period of vibration of any high-rise building through idealization as a fixed-ended cantilever with lumped masses.

If the response is measured at point B in direction x (for example), for an excitation at point A in direction y, then the transfer function (crudely Bx/Ay in the frequency domain) is identical to that which is obtained when the response at Ay is measured when excited at Bx. That is Bx/Ay=Ay/Bx. Again this assumes (and is a good test for) linearity. (Furthermore, this assumes restricted types of damping and restricted types of active feedback.) (*Modal Analysis*, n.d.)

MODAL ANALYSIS USING FEM (FINITE ELEMENT METHOD):

The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration. It is common to use the finite element method (FEM) to perform this analysis because, like other calculations using the FEM, the object being analysed can have arbitrary shape and the results of the calculations are acceptable. The types of equations which arise from modal analysis are those seen in eigensystems. The physical interpretation of the eigenvalues and eigenvectors which come from solving the system are that they represent the frequencies and corresponding mode shapes. Sometimes, the only desired modes are the lowest frequencies because they can be the most prominent modes at which the object will vibrate, dominating all the higher frequency modes.

It is also possible to test a physical object to determine its natural frequencies and mode shapes. This is called an Experimental Modal Analysis. The results of the physical test can be used to calibrate a finite element model to determine if the underlying assumptions made were correct (for example, correct material properties and boundary conditions were used). (*Modal Analysis using FEM*, n.d.)

MECHANICAL RESONANCE

Mechanical resonance is the tendency of a mechanical system to respond at greater amplitude when the frequency of its oscillations matches the system natural frequency of vibration (its *resonance frequency* or *resonant frequency*) than it does at other frequencies. It may cause violent swaying motions and even catastrophic failure in improperly constructed structures including bridges, buildings and airplanes. This is a phenomenon known as resonance disaster.

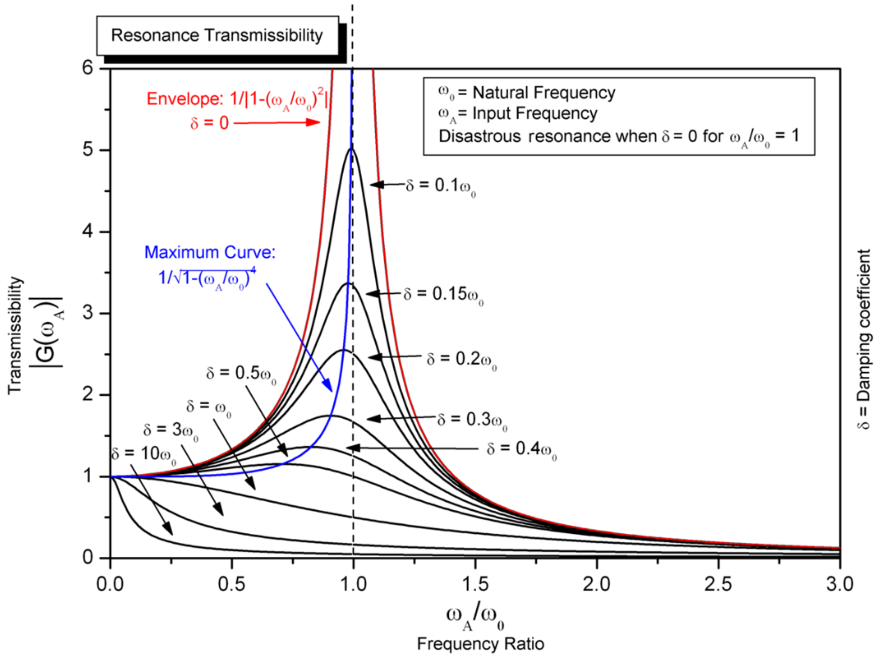


Figure 1. Graph showing mechanical resonance in a mechanical oscillatory system

Avoiding resonance disasters is a major concern in every building, tower and bridge construction project. The Taipei 101 building relies on a 660-ton pendulum—a tuned mass damper—to modify the response at resonance. Furthermore, the structure is designed to resonate at a frequency which does not typically occur. Buildings in seismic zones are often constructed to take into account the oscillating frequencies of expected ground motion. In addition, engineers designing objects having engines must ensure that the mechanical resonant frequencies of the component parts do not match driving vibrational frequencies of the motors or other strongly oscillating parts.

Many resonant objects have more than one resonance frequency. It will vibrate easily at those frequencies, and less so at other frequencies. Many clocks keep time by mechanical resonance in a balance wheel, pendulum, or quartz crystal.

The natural frequency of a simple mechanical system consisting of a weight suspended by a spring is:

$$f = \frac{1}{2\pi}\sqrt{\frac{k}{m}}$$

Where *m* is the mass and *k* is the spring constant.

Mechanical resonators work by transferring energy repeatedly from kinetic to potential form and back again. Some resonant objects have more than one resonance frequency, particularly at harmonics (multiples) of the strongest resonance. It will vibrate easily at those frequencies, and less so at other frequencies. It will "pick out" its resonance frequency from a complex excitation, such as an impulse or a wideband noise excitation. In effect, it is filtering out all frequencies other than its resonance. In the example above, the swing cannot easily be excited by harmonic frequencies, but can be excited by subharmonics. (*Mechanical Resonance*, n.d.)

PARTICULARITIES ASSOCIATED TO THE PROCESS STUDIED

Coming back to the case we are studying, to make the small cylinder start vibrating and not just getting deformed from the vibration of the press, it is important to consider two different things:

- The piece has to move on the elastic behaviour. If each deformation due to the vibration creates some deformation, the elasticity compound, will be less useful.
- The created vibration has to have a frequency very close or equal to the resonance one.
- It is not enough vibration movement. Vibration cause some important deformation, that is why it is also necessary to include an advance in the movement to make the distance between bodies keep constant and do not increase making the load minor.

MICROFORMING

Microforming is a well suited technology to manufacture very small metallic parts, in particular for mass production, as they are required in many industrial products resulting from microtechnology. Compared to other manufacturing technologies microforming features specific economic and ecological advantages. Nevertheless, there are only some singular applications known until today. This paper tries to find out the reason why, analysing systematically the problems emerging in transferring the know-how on forming from the macro- to the microworld. Reviewing the state of the art in basic and applied research reveals that scaling effects do appear not only within the process but must be taken into account in all the other areas of the whole forming system as well, demanding finally new solutions especially for tool manufacturing and machine concepts. Recent progress, innovative ideas and new developments on these sectors represent a promising basis to exploit the inherent potential of microforming in the future. (Geiger et al., 2001)

MICROFORMING PROBLEMS

The problems appearing currently in microforming which also complicate further steps in miniaturization are obviously strongly coupled with miniaturization itself. In order to come to an approach for a better understanding of these effects, it is useful to consider the microforming system split up into the four groups (Figure 4):

- Material
- Process
- Tools and
- Machine/equipment.

In all these areas specific effects of miniaturization are observed. The material behaviour is influenced by size effects that occur when scaling down a process from conventional size to the micro scale. The flow stress, anisotropy, ductility and the forming limit depend on the specimen size and the microstructure, which has to be considered when designing a micro forming process. The process is of course strongly coupled with the material. Thus, material effects influence the process. However, there are additional effects concerning the forming forces, tribology, spring-back, the scatter of the results and thus the accuracy of the parts to be produced. It should be noted that the effects of material and process also affect the applicability of FEM-based simulations for process design and layout.

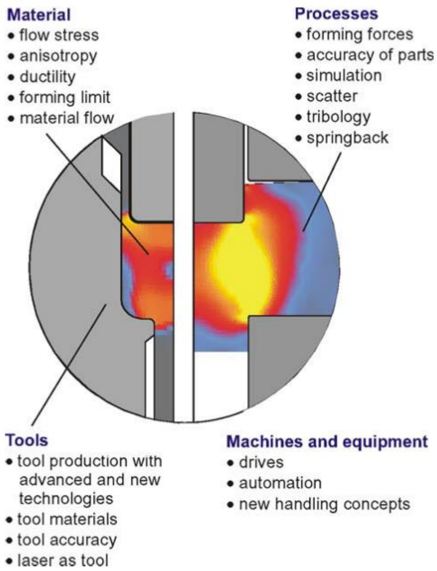


Figure 2. Problems in the microworld (LFT)

With respect to tooling systems the main problem lies in the manufacturing of high precision tools and the availability of adequate machine tools for micro forming operations. Especially the manufacturing of complex inner shapes of extrusion dies with close tolerances and a sufficient surface quality is difficult and up to now limited to a minimum diameter of 0.5 mm. However, new approaches involving alternative manufacturing methods exist to overcome these difficulties.

An essential problem for micro machines and components is evidently the required precision at high speeds. A machine that produces 300 parts per minute with a diameter of 0.5 mm has to transfer the part between the forming stations within less than 0.2 seconds and position it in or above the die with an accuracy of a few microns. The surface where the part can be gripped is extremely small and the part weight too low to overcome adhesion forces.

Also the clearance or backlash between drive and punch of a regular forming machine, that is negligible for a conventional specimen size, can be a problem, when the total required stroke to form a micropart lies in the range of a few 100 pm. Additionally, adequate measurement technology is required in order to ensure the product quality and to enable a process control. Finally, the extremely small part dimensions might make it necessary to produce in clean rooms, which is cost-intensive and must lead to new machine concepts.

(W. Presz, 2019, 2020; Wojciech Presz, 2018, 2019; Project, 2020)

ABOUT MARC MENTAT PROGRAM

First of all, it is obligatory to mention that the version used in this project has been the student version of the Marc Mentat program property of MSC software. This student version has several limitations. Some of them are listed below.

- Restricted number of finite elements to include in the simulation. This is not a real problem in the simulations that are going to be built in this part of the general problem because working on two-dimensions allows to use a much lower number of these elements. Anyway, with a wider range of elements, results would be more accurate.

- One of the most important restrictions is the limitations in the possible frequency associated to the punches. This fact make impossible to associate the resonance frequency to the punch in each particular case.
- There are also some few problems with the representation of the simulation once it has been submitted. This problem is quite important because if the simulation can run but results cannot be observed, the simulation is worthless. This fact happens, as previously commented in high frequencies, with similar strength of materials tested and in some other specific cases.

All these things considered, for the main part, the modal analysis, another program will be used, and its name is Ansys. Marc Mentat allows to do this kind of analysis, or at least it appears the loadcase options, but at the moment of truth, this is not possible or I have not been able to make it run.

ANALYSIS OF THE RESULTS – STATIC MODEL WITH DIFFERENT PUNCH SHAPES

Firstly, some calculations will be done in Marc Mentat program. First simulations (in static mode) are made in order to know the final shape that the cylinder will have and to get familiarize with the vibrations that will be subsequently treated and with the program. These probes will also help to understand why the loss of contact between sonotrode and shape affects. This effect, in reality, do not happen in a controlled way, it happens uncontrollably due to the increase in the wave amplitude along the sonotrode due to the frequency and energy preservation.

PROCEDURE FOLLOWED TO CREATE THE FINAL VIBRATION TABLE:

Prior to the main analysis, a short introduction about how to make punches to get approach to the tops of the cylinder shape to make the simulation more efficient and logical in order to maximise simulation time.

To begin with, the table implemented in each punch in the scenarios with 2 punches instead of a base and a punch is going to be the same but with the only difference that one of them will have a multiplier (-1) when implementing this position table to change the direction of the movement.

Characteristics associated to the loadcase implemented are:

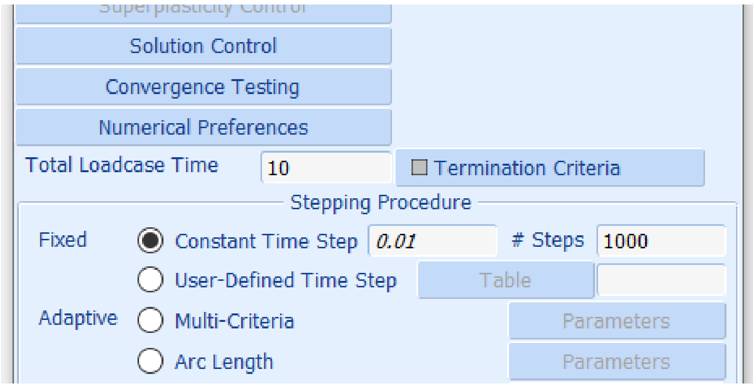


Figure 3. Loadcase characteristics

This means that 1000 different calculations will be done to analyse the process in order to have a quite exact result.

INITIAL APPROACH CALCULATION

This approach is studied independently because will be used for all simulations in the same way.

Distance calculation from the press to the cylinder to tart the compression:

Nearer point → the cylinder corner

Current x axis distance from that corner point to the origin of coordinates = 0.5 mm (half total height of the cylinder)

Current x axis distance from the first press part that is going to get in touch with this corner:

$$Distance\ in\ x\ axis = \sqrt{0.8^2 - 0.5^2} = 0.624499 \approx 0.6245\ mm$$

Consequently the distance that the press has to advance before starting with the vibration is:

$$0.6245 - 0.5 = 0.1244\ mm$$

Once the distance to be traversed is known, it is time to adjust the velocity that this presses or punches have to be associated with, or the same way, adjust the position along simulation time (time is not conceived as we think, it is just a variable to work with, but on static mode, time is not existing).

In case of using the velocity possibility of the tables associate to the bodies properties, it would be (and constant):

$$Planar\ cases\ \rightarrow \frac{0.1\ mm}{0.8\ sec} = 0.125\frac{mm}{sec}$$

$$(in\ order\ to\ leave\ 2\%\ of\ wait\ till\ the\ begining\ of\ the\ vibration\ when\ 10\%\ will\ be\ finished)$$

$$Curved\ punch\ case\ \rightarrow \frac{0.1244\ mm}{0.8\ sec} = 0.1555\frac{mm}{sec}$$

$$(same\ purpose)$$

On the case that is going to be implemented, the position is calculated, actually by different trial on the Excel program and changing on it the different parameters as frequency, amplitude and so on, but for the beginning, just the approach is going to be explained:

Starting point (obviously) begins in the defined layout position so in the position/velocity graph, this point is going to be represent in the 0. To approach to the part a constant velocity approach is going to be used, as the position is linearly increased till a very short distance is achieved, then the vibration will be able to start.

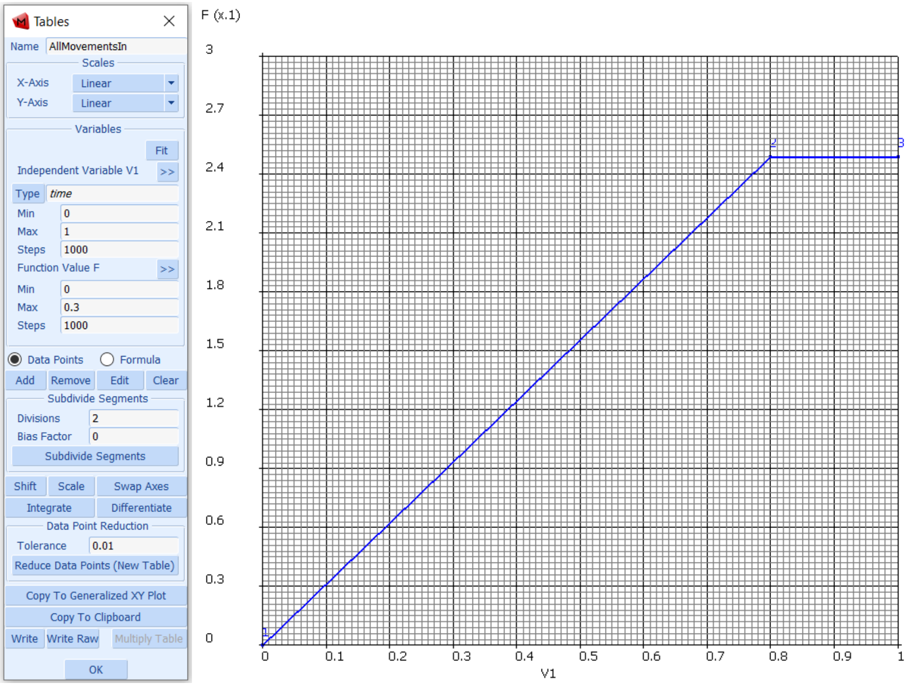


Figure 4. Approach graph

It is also important to notice that steps have been chosen equal to 1000 in both axis in order to have a much better accuracy than with a lower number. It is possible because the submission time is not too long even with this modification.

As it can be observed, in this 10 seconds simulation, at the beginning, 0.8 seconds are going to be dedicated to the approach and following 0.2 seconds there will not be any movement of the press/presses. Next the vibration will start to the second 9 of the simulation, when the leave will start to the starting position.

VIBRATION

The goal of the vibration is to know the response of the cylinder part to some changes in the punch and base shapes. To perfectly know the differences of these changes, it is important to keep the vibration parameters, that is why the amplitude, phase, frequency and advance have not been changed along the results later mentioned.

Another important fact to take into account is the phase associated to the sin. This is because the beginning of the vibration has to be in the higher part of the sin, otherwise the first hit of the punch would cause an excessive deformation, and that is not what we are looking for.

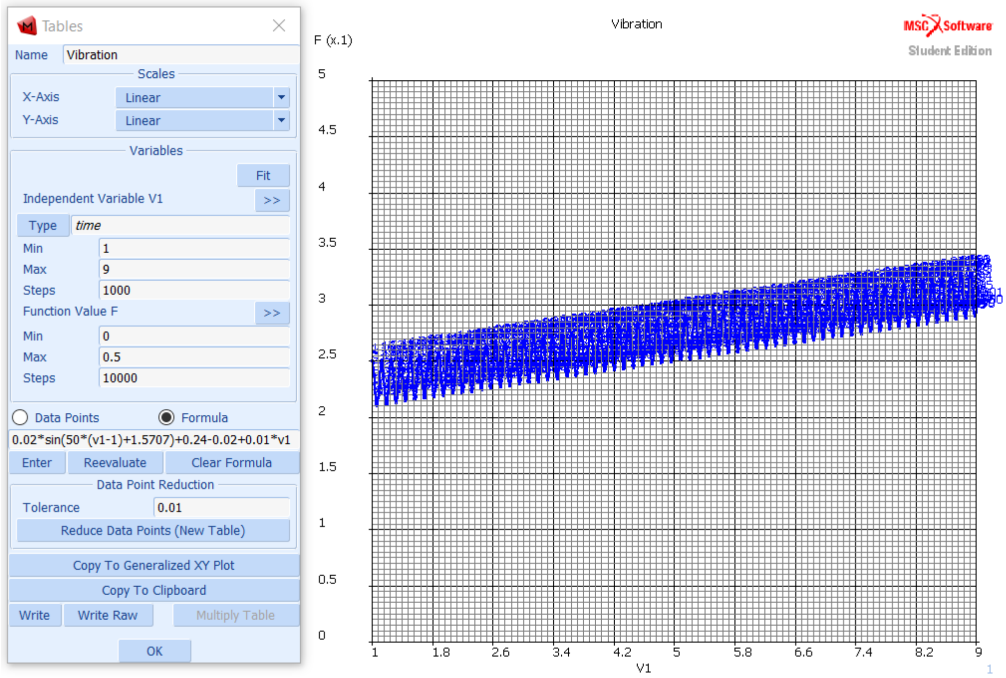


Figure 5. Vibration implemented to the movement graph

On the picture above it is observable the amount of the periods of the wave associated to the frequency chosen, in these cases f = 50 Hz. Below the zoom made to the beginning of this wave to observe the beginning of it, being in the upper peak.

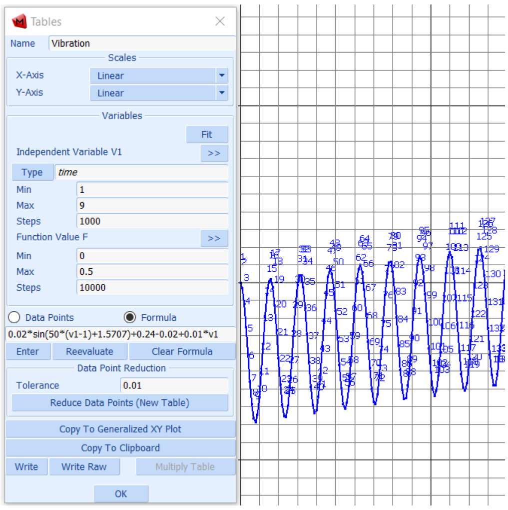


Figure 6. Beginning zoom of previous variation

Formula used and its breakdown:

$$0.02 * \sin(50 * (v1 - 1) + 1.5707) + 0.24 - 0.02 + 0.01 * v1$$

$$Amplitude \ \rightarrow 0.02$$

$$Sin\ frequency \rightarrow 50\ Hz$$

$$Independent\ variable\ + starting\ from\ 0\ factor\ \ \rightarrow v1 - 1$$

$$Phasae\ \rightarrow \frac{\pi}{2} = 1.5707\ rad$$

$$2\ adjustments\ factors\ \rightarrow$$

$$\rightarrow 0.24\ (adding\ the\ initial\ approach) - 0.02\ (minimicing\ the\ advance)$$

$$Advance\ long\ time \rightarrow 0.01 mm/sec$$

Finally the following final table is obtained:

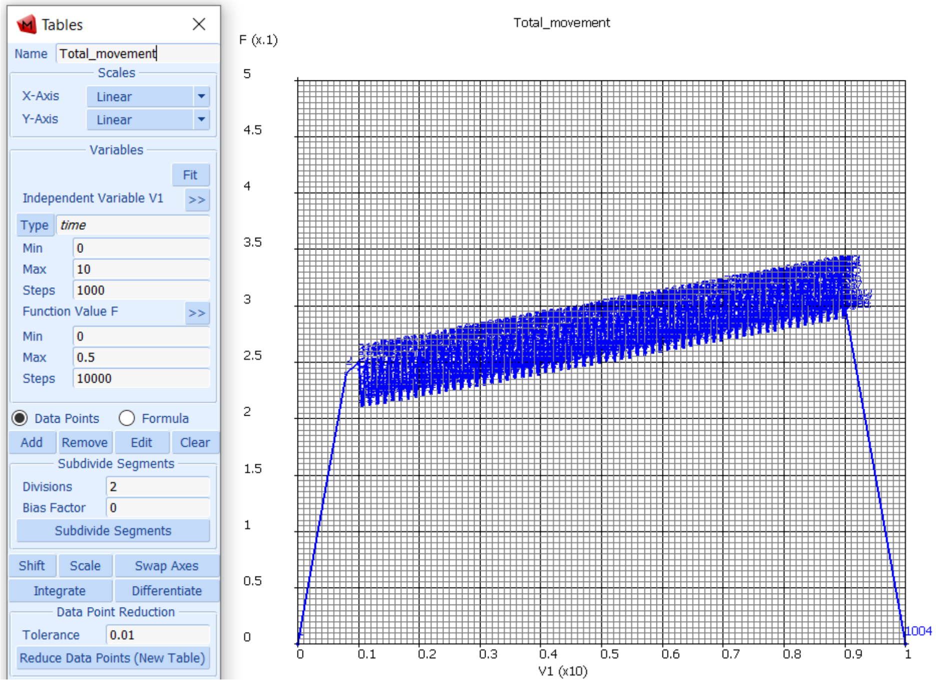


Figure 7. Total punch movement graph

The vibration and adjustments is exactly the same for the planar punches but with the only difference that the initial and final approaches are not equal to 0.1244 but 0.1 mm (the starting gap between the cylinder and each punch.

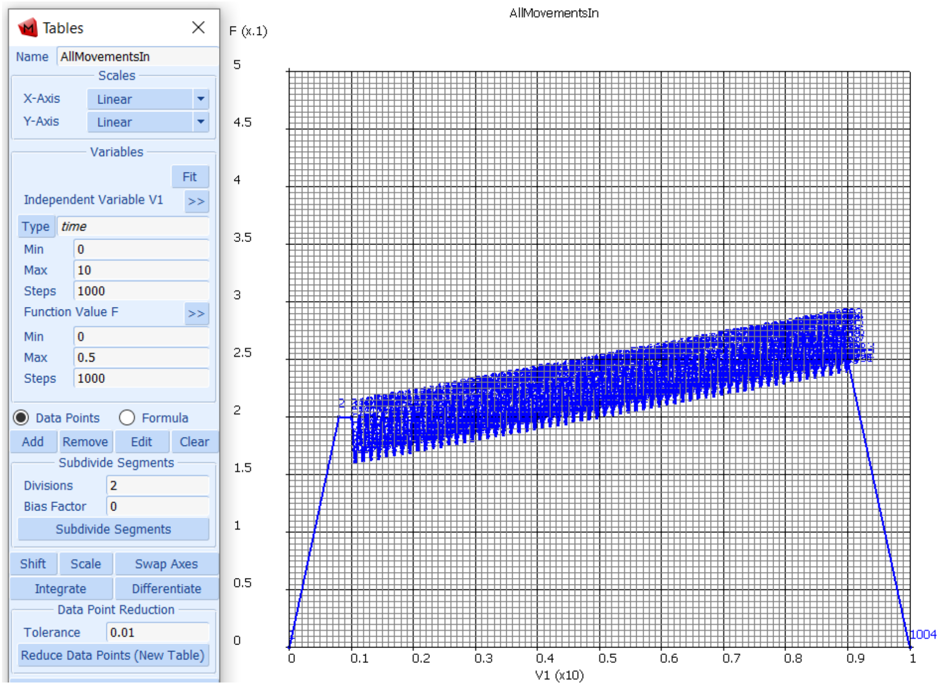


Figure 8. Total movement graph for planar punches

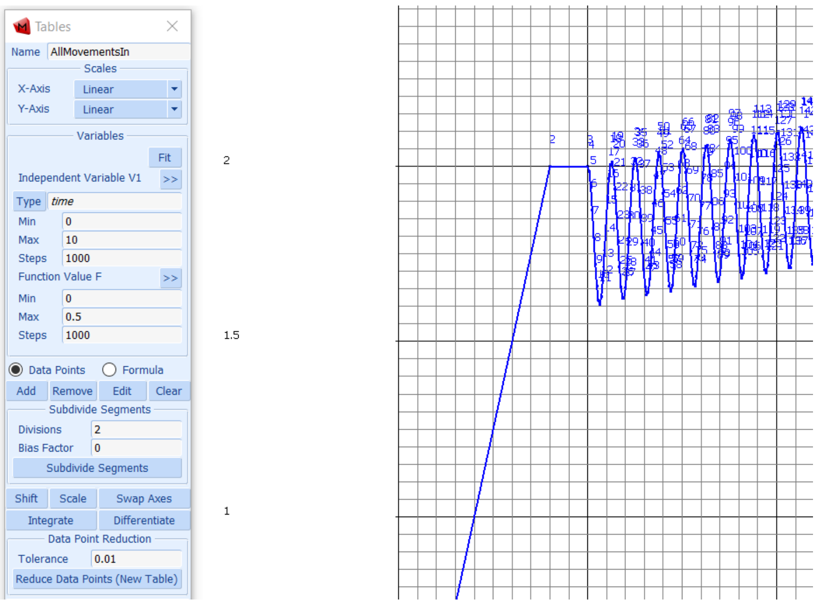


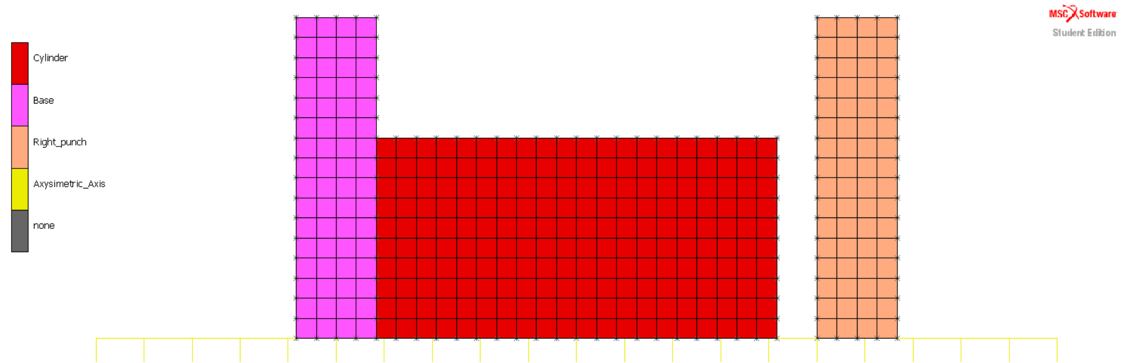
Figure 9. Beginning zoom of the previous planar punch vibration

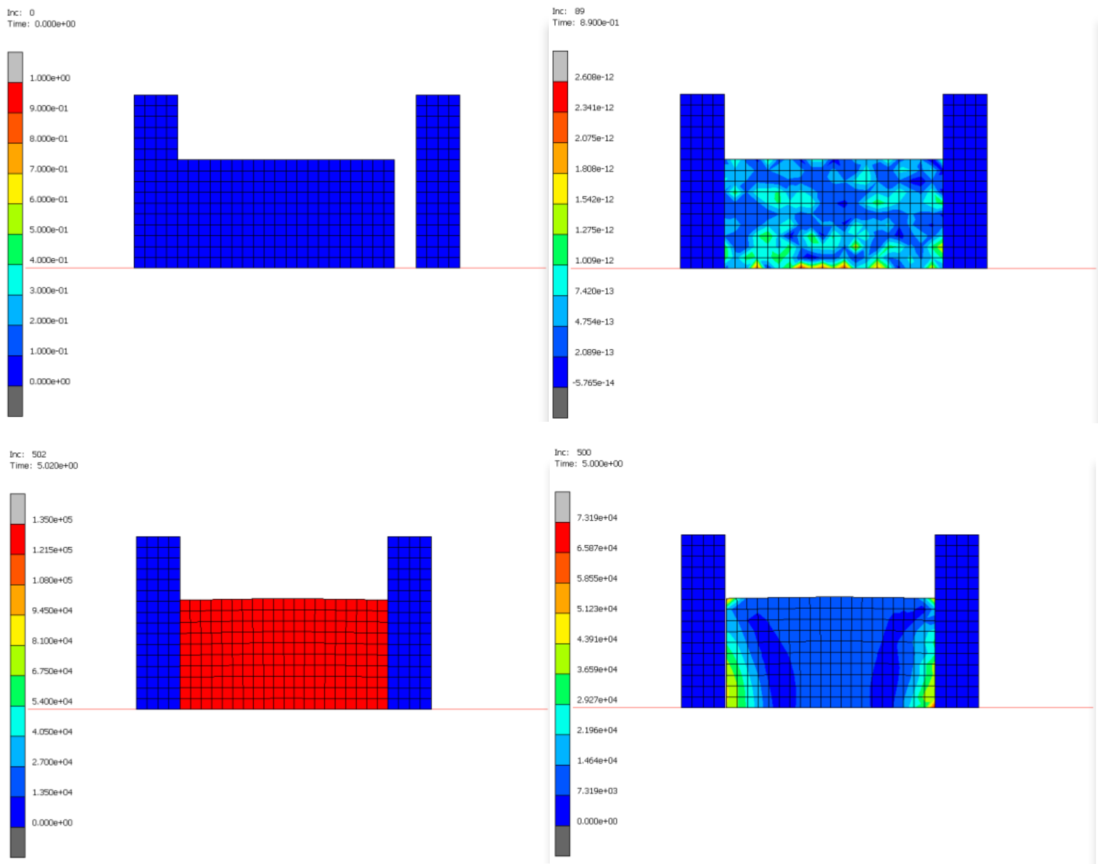
Once the punch trajectory has been developed, it is time to focus on the results obtained in the 4 different static simulations.

All simulations of the four following cases are going to be made with aluminium as material for the cylinder and on next point this material will be changed in order to get the differences occurred by changing the material studied. The purpose of changing these punches shape is trying to find a process similar to the phenomenon that really happens with the cylinder shape. These different simulations are carried out trying to find which probe gives a more similar shape result according to the final cylinder form that we want to achieve, as in the other part of the work carried out by Ignacio. This will also help me to get familiarize with the program and the vibration modes.

1 PLANAR PUNCH AND A PLANAR BASE

First and most simple case is a planar base which supports the cylinder in the proper position and a planar punch in change of transmitting the vibration movement to the shape and the loads created due to this movement.





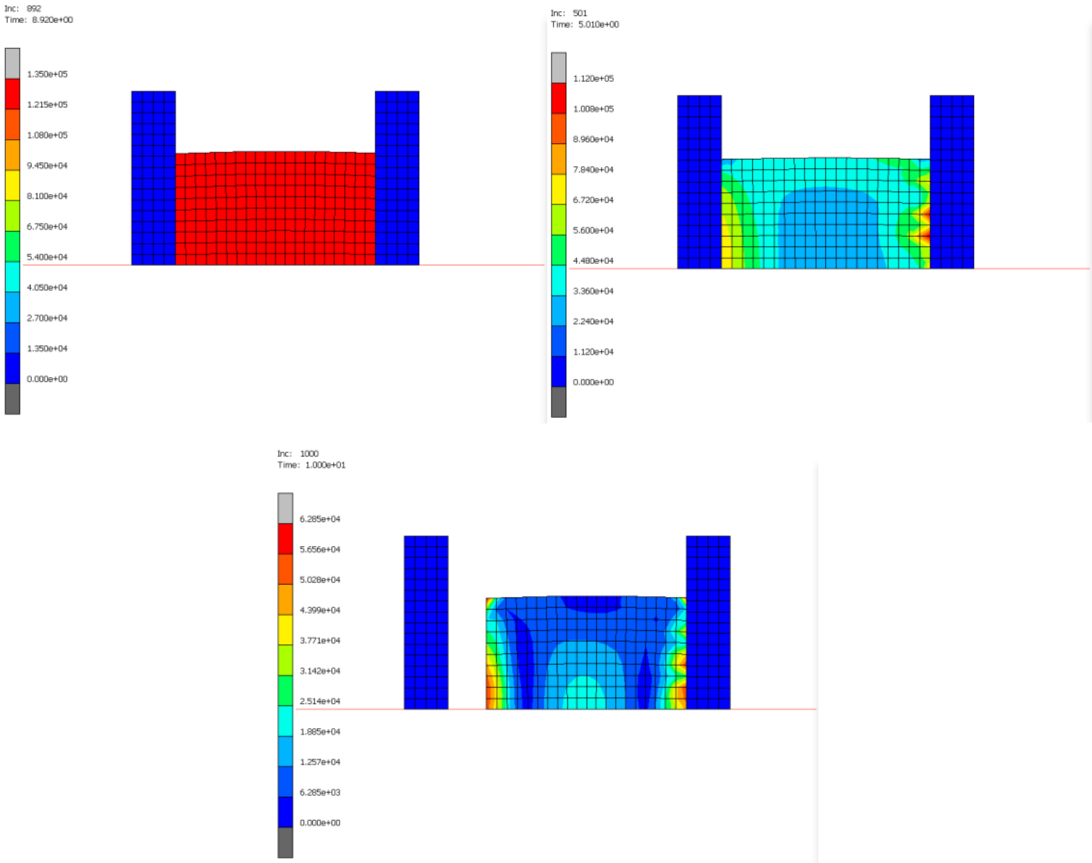
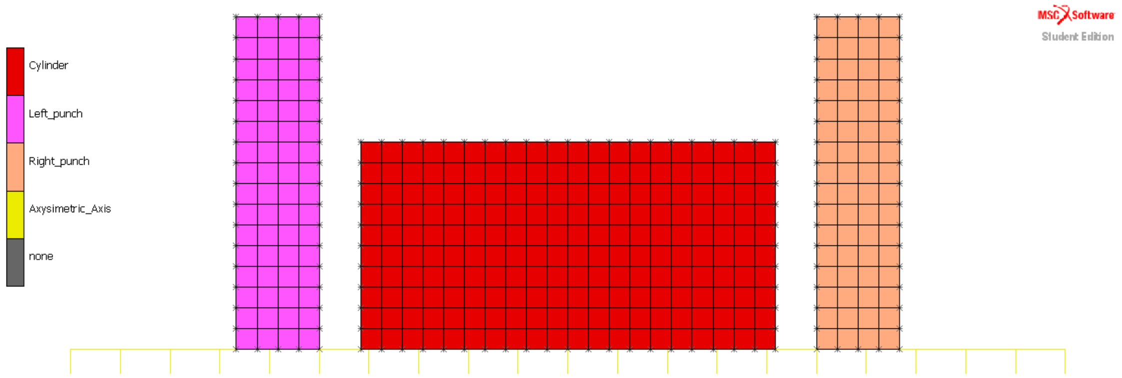


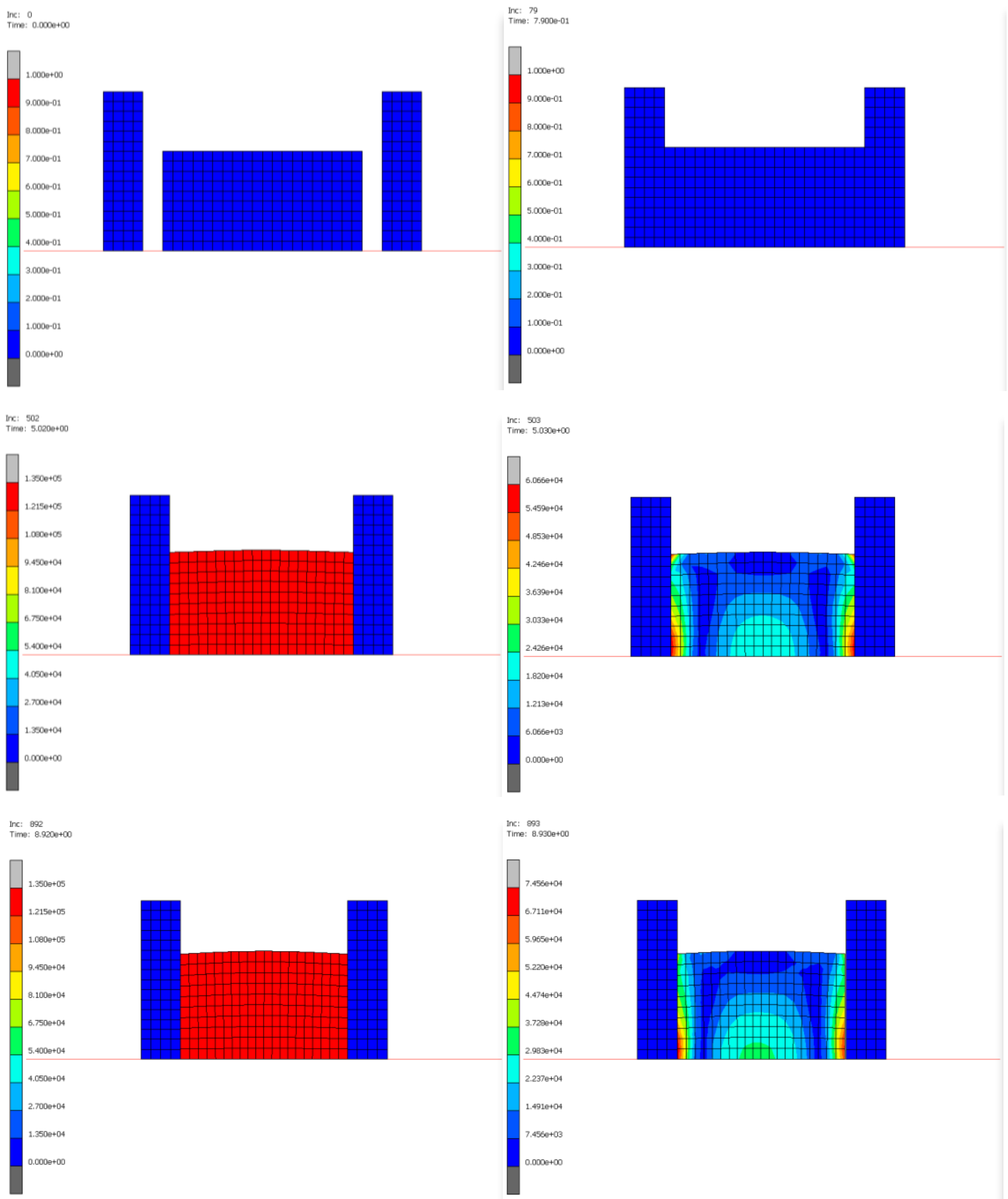
Figure 10. Graphs of planar base and planar punch simulation

By analysing this results it is easy to see that Equivalent von Misses stress is maximum on the extremes of the cylinder faces due to the friction with the punch and base and this value is equal to 1.350e+05 when the load is active. When this load is not active, what means to be in the valley of the vibration movement, the residual maximum von Misses stress is 7.319e+04 in the middle of the process and it reached 1.125e+05 just after the last push has been done. Final maximum residual von Misses stress once the punch has been separated to the starting position is 6.285e+04, almost half of the stress happened in the last analysed case.

2 PLANAR PUNCHES

On this new case, the base, before positioned on the left of the shape in the representation of the simulation, is going to be replaced by another punch which will have the same movement than the right one but in opposite direction, so the load will be provided at the same time for both punches, creating by this way the double of stress than in previous case. Relativizing movement, it is like having the same movement as previously but with the double of amplitude and advance of the punch, in other words, we have the same case but with double load as before.


MSC Software
Student Edition
Cylinder
Left_punch
Right_punch
Axysimetric_Axis
none


Inc: 0
Time: 0.000e+00
1.000e+00
9.000e-01
8.000e-01
7.000e-01
6.000e-01
5.000e-01
4.000e-01
3.000e-01
2.000e-01
1.000e-01
0.000e+00
Inc: 79
Time: 7.900e-01
1.000e+00
9.000e-01
8.000e-01
7.000e-01
6.000e-01
5.000e-01
4.000e-01
3.000e-01
2.000e-01
1.000e-01
0.000e+00
Inc: 502
Time: 5.020e+00
1.350e+05
1.215e+05
1.080e+05
9.450e+04
8.100e+04
6.750e+04
5.400e+04
4.050e+04
2.700e+04
1.350e+04
0.000e+00
Inc: 503
Time: 5.030e+00
6.066e+04
5.459e+04
4.853e+04
4.246e+04
3.639e+04
3.033e+04
2.426e+04
1.820e+04
1.213e+04
6.066e+03
0.000e+00
Inc: 892
Time: 8.920e+00
1.350e+05
1.215e+05
1.080e+05
9.450e+04
8.100e+04
6.750e+04
5.400e+04
4.050e+04
2.700e+04
1.350e+04
0.000e+00
Inc: 893
Time: 8.930e+00
7.456e+04
6.711e+04
5.965e+04
5.220e+04
4.474e+04
3.728e+04
2.983e+04
2.237e+04
1.491e+04
7.456e+03
0.000e+00

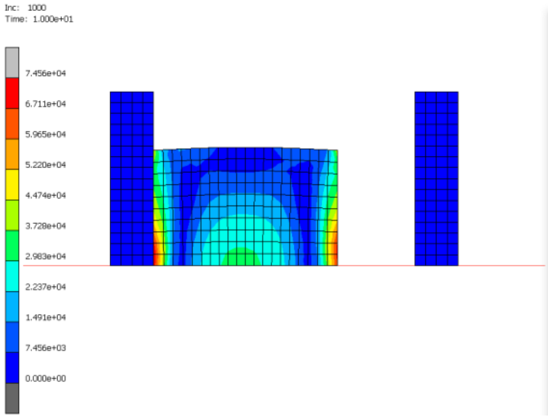


Figure 11. Graphs of two planar punches simulation

On this case, results are a bit different. Stress during compression is 1.350e+05 both in middle of the simulation or at the end of it and stress when loads are not active are 6.066e+04 in the middle of this case and 7.456e+04 (a bit higher) at the end of the simulation. In this case, this value do not change when the punches go back to their initial state, what did not happen in the previous case with one base and one punch.

1 CURVE PUNCH AND A PLANAR BASE

Coming nearer to the real case it is wanted to be studied, now this curve punch is implemented with, in this first case, a planar base on the bottom of the shape. Now initial approach is even more important because the initial touch will be just stood by the perimeter of the circle which is the front face of the cylinder. This approach has already been explained.

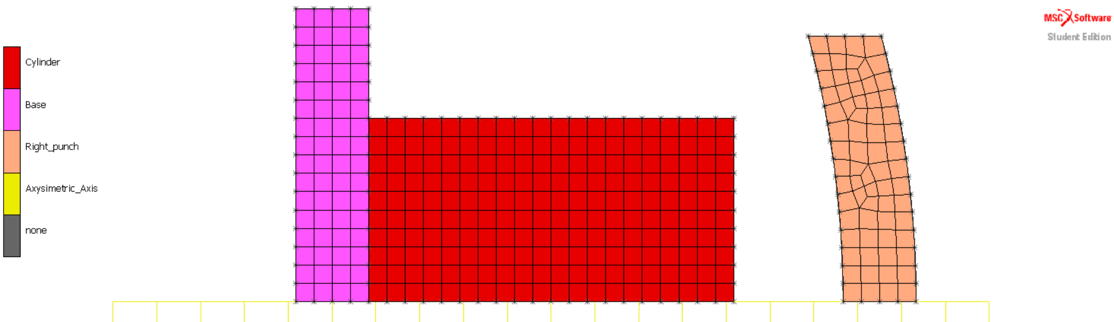
The characteristics of the punch are as follows:

$$Angle\ represented\ (not\ important) \rightarrow \theta = 15\ º$$

$$It\ is\ only\ important\ that\ it\ is\ longer\ than\ the\ cylinder\ diameter$$

$$Radius\ (very\ important)\ \rightarrow R = 2.8\ mm$$

Now the analysis starts:



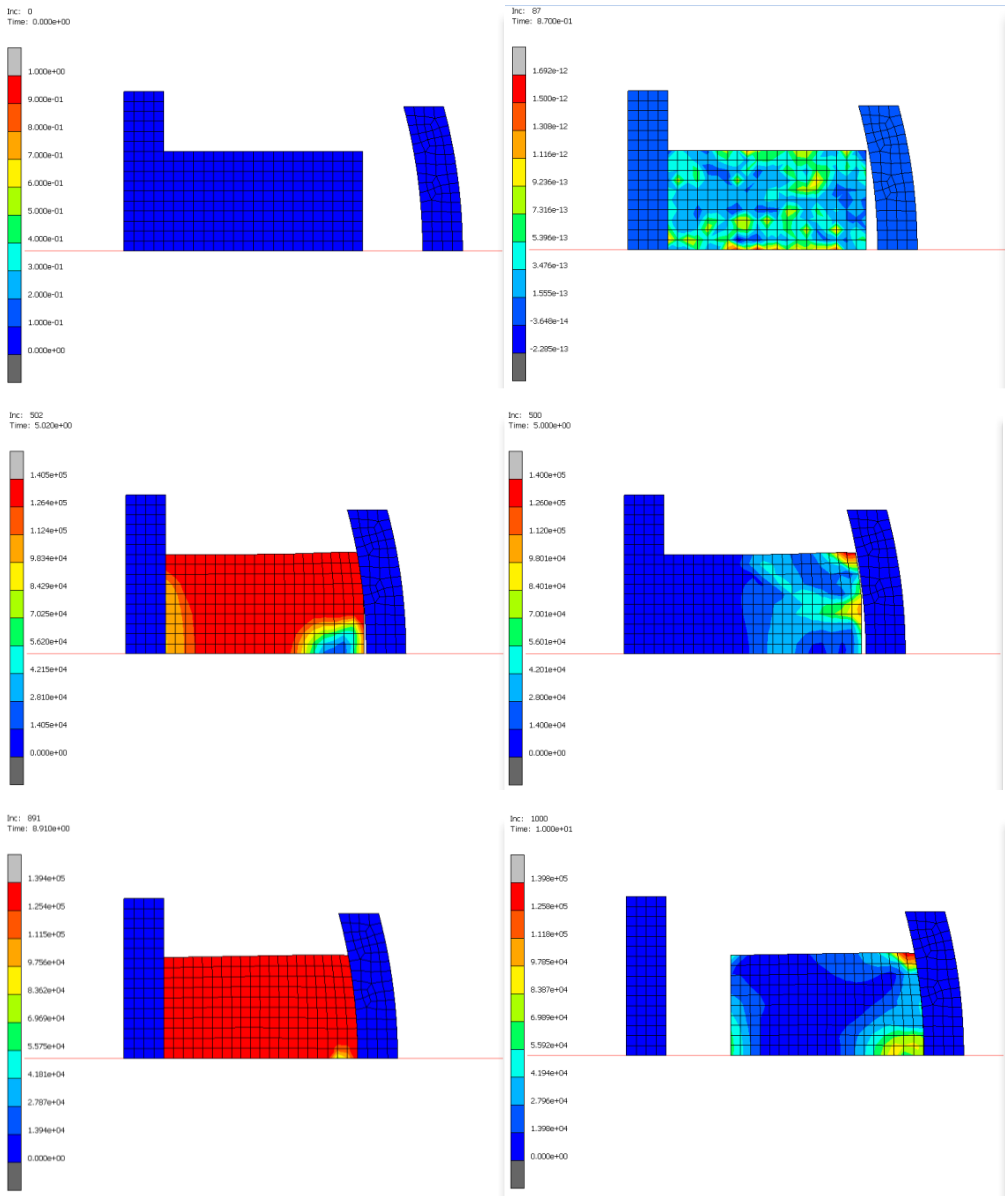


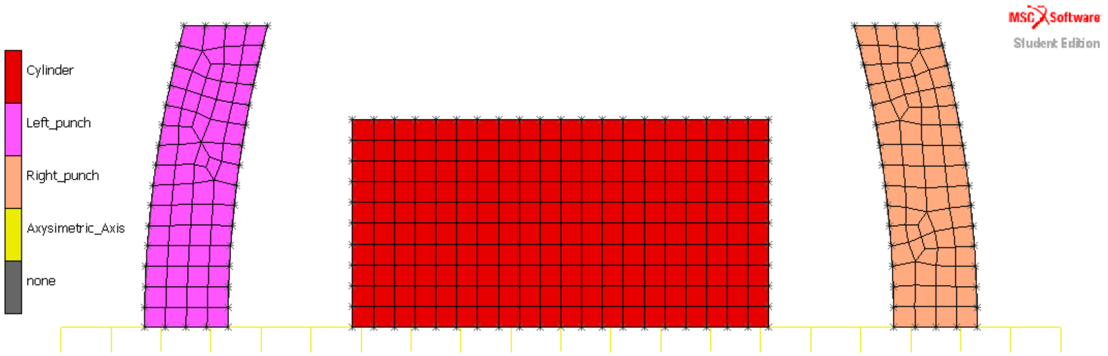
Figure 12. Graphs of planar base and curved punch simulation

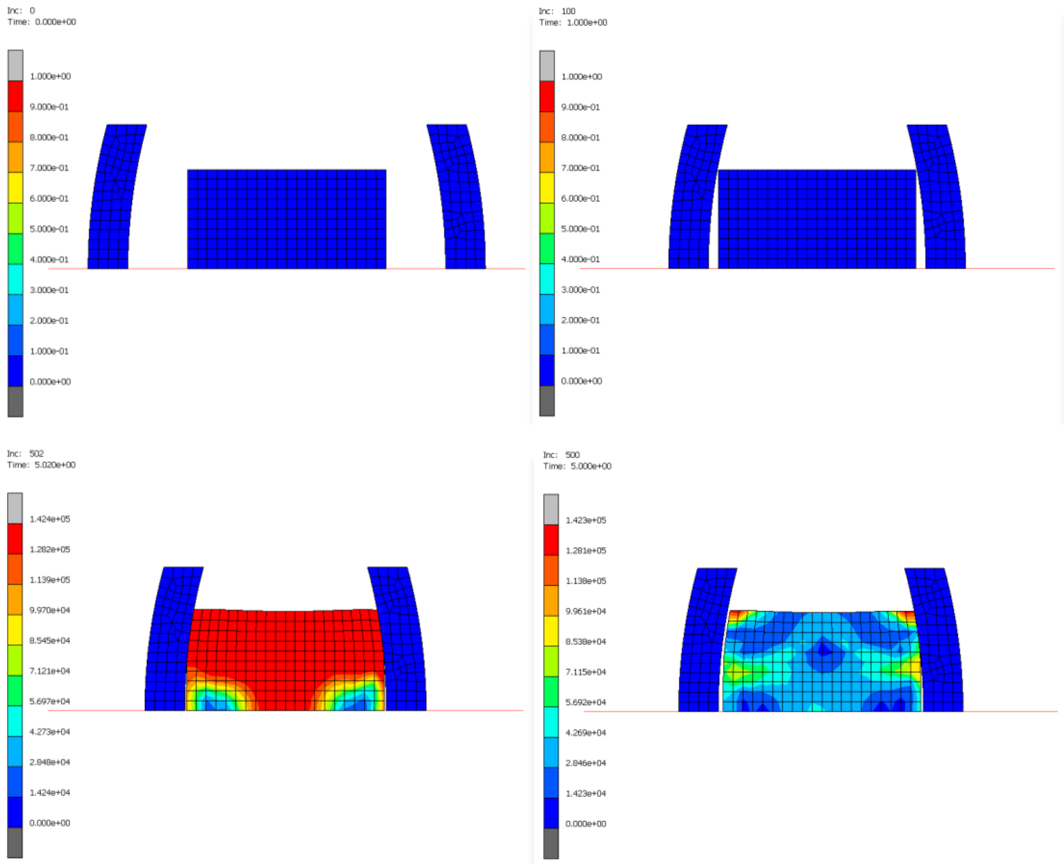
In this case, the von Misses stress in the part is quite similar when the load is active or inactive in the middle of the process with values equal to 1.405e+05 and 1.400e+05 respectively and still not a total contact of the punch in the front surface. As it is possible to observe, this is the only case where the external surface of the cylinder is not symmetrical due to the difference in the two punches shapes (actually base and punch).

At the end of the simulation, maximum von Misses stress is 1.398e+05, more than in previous cases cause of this change in the punch shape. It is also important to realise the differences in the stress distribution visible in the colour bands of every image.

2 CURVE PRESSES

Final and nearer case to the reality that is being studied on this project related to the ultrasonic orbital microforming is the two curved punches and following results.





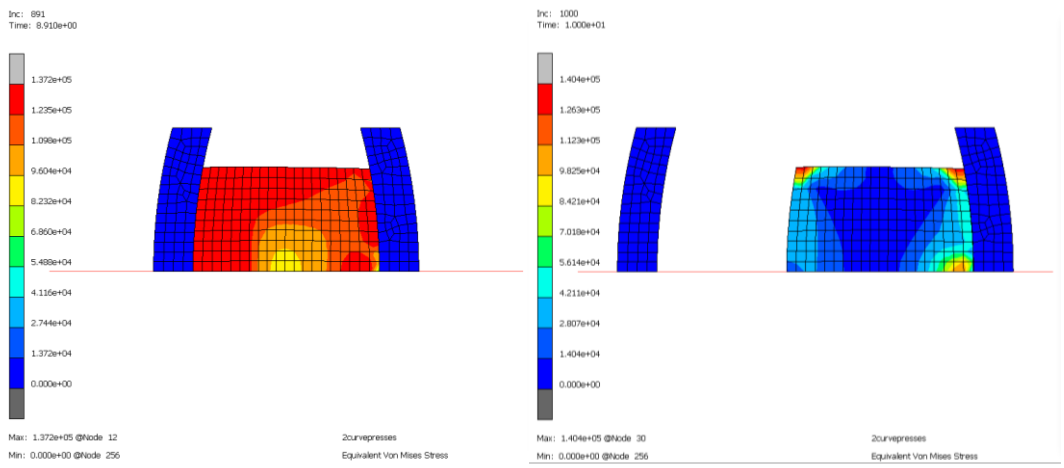


Figure 13. Graphs of two curved punches simulation

As an important aspect to comment, there is not still a total contact with both surfaces when the simulation has already be run till 50% and by this time the von Mises stress is already very high, with a maximum value equal to 1.424e+05 when both punches have active their load (upper peak of the vibration wave). The maximum stress value at the end of the analysis is 1.404e+05 once the load has been removed and the punches have gone back to their initial position. Again a higher value than the previous cases, mainly due to the punches shape.

ANALYSIS OF THE RESULTS – STATIC MODEL WITH DIFFERENT CYLINDER MATERIALS WITH TWO CURVED PUNCHES

Once we have found the test that better matches the results we are looking for, it is time to analyse the results associated with three materials, in our case, three of the most common structurally speaking, aluminium, copper and steel. This results are going to be analysed with the two curved punches because of, despite the fact that punches are not this way at all, the effect caused on the small part is the same as if they are curved.

Starting from the images that represent the expansion of the 2D simulation that is going to be developed by changing the cylinder material from Aluminium, Copper and Stainless Steel to analyse the difference in the internal stress suffered by the part when the same punches movement is provided in three cases.

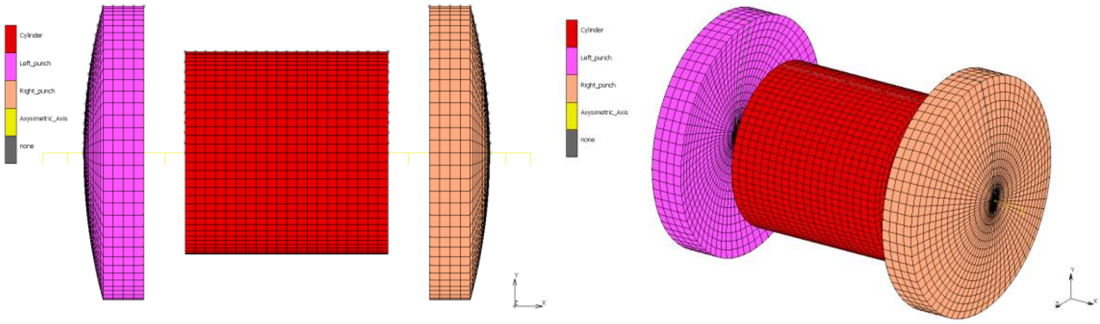


Figure 14. 3D analysis representation

From now on, the analysis is going to be focused on each material by separated and after the three analysis, a comparison among them will be done. In each different part, the stress distribution is going to be studied associated with the colour bands draw and the displacement in axis Y, which represents the expansion along the radius of the cylinder. Complete analysis is developed in the Appendix 1 - DIFFERENT MATERIALS SIMULATION RESULTS. Here just graphs with forces along the simulations are shown:

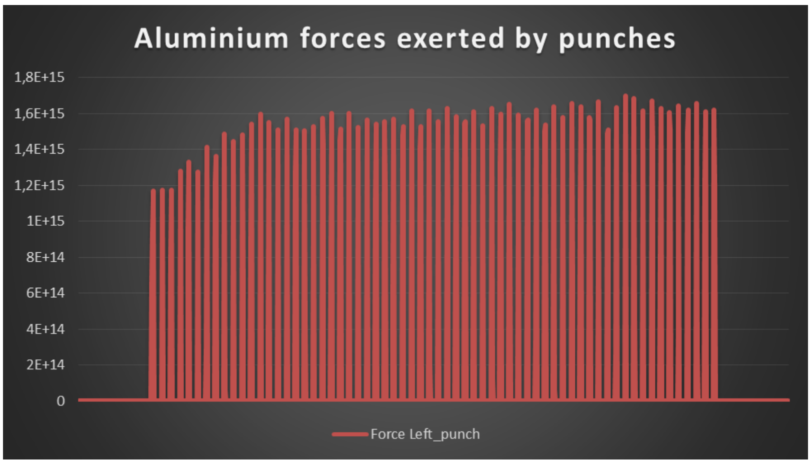


Figure 15. Aluminium forces exerted by punches

It is only represented the force exerted by the left punch because it is the same than the force experimented by the part, as a reaction of this one, and very similar to the force carried on by the right punch. As we can observe, this force starts from a linear growth to few moments after the beginning till the end of the compression follows a very stable trend. The maximum load experimented is a bit higher than 1,6 x 10^{15}.

As a clarifying commentary it is important to mention that this graph is showing in red areas when the punch is pressing the part and with no colour when there is gap between part and punch (punch is not active or, in other words, it is in the down part of the vibration wave).

Going directly to the second material, the results are shown below:

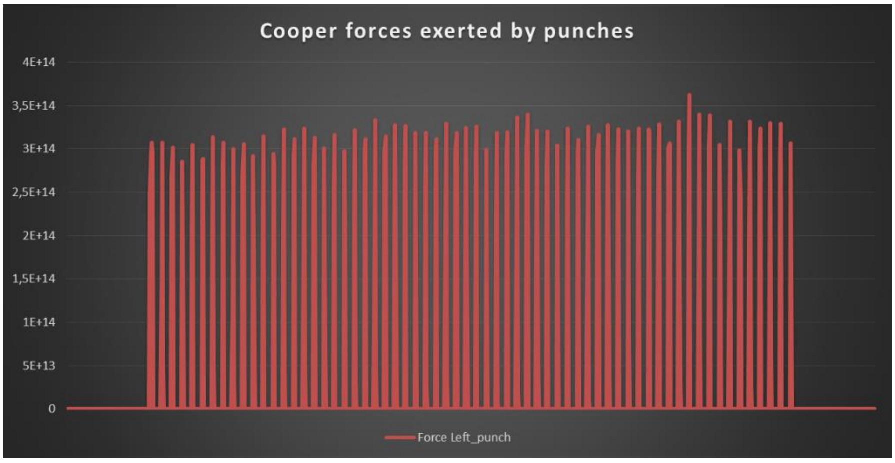


Figure 16. Cooper forces exerted by punches

On this new material (Cooper) the force distribution is more uniform. The average module of this force is around 3 x 10^{14} but the higher value exceeds the 3,5 x 10^{14} mark. Force experimented on the punch is much lower than in previous case due to the lower Tensile Yield Strength of this material in relation with the Aluminium forces.

Finally, Steel is going to be analysed:

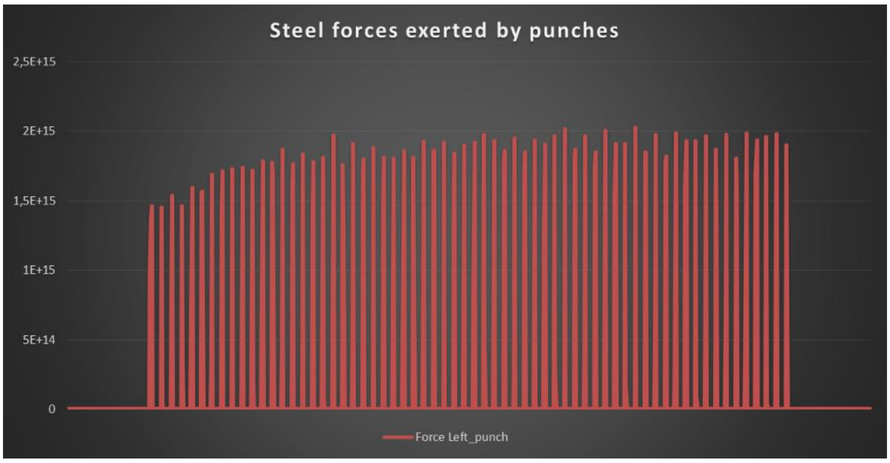


Figure 17. Steel forces exerted by punches

Again it is observed a ramp at the beginning as it happened with the Aluminium. This is possibly because these two materials, despite their differences (Steel is obviously stronger than Aluminium, and this second one is softer), the have similar behaviours. This material will experiment a maximum force equal to more than 2 x 10^{15}. This is the highest force of all three materials and trend is again quite stable despite the change in the shape as it can be observed in the appendix.

COMPARISON AMONG MATERIALS

As a final conclusion sharing of the different results obtained, it is remarkable:

- Forecasted stress needed to deform each material was correct. The necessary stress is a bit higher than the Yield Stress associated with each material in particular.
  That is why load needed are: Cooper < Aluminium < Steel, according to the mentioned Yield Stress.
  Also important to visualise the similar behaviour between Aluminium and Steel despite their properties differences.
- When the punches are in their upper peak of their respective waves, the material is suppressed to its maximum stress and it gets deformed. When this fact happens, all the cylinder is under the same load and it is equal to a value a bit higher than its Yield Stressed as commented in previous point.
- Steel and Aluminium have a similar behaviour despite of the differences in stress module either in deformation or in residual stresses. It is quite remarkable in the colour bands of each figure. After stress is applied, Aluminium and Steel parts have the highest load at their outside corners and it blurs through the surfaces, meanwhile in the heart of the cylinder there is not any residual stress. On the other hand, in the Cooper shape, the stress distribution inside the part when the loads are not acting has a very different draw, much more abstract and without following a specific patter.
- Same as point above happens with the Y axis displacements figures. The Aluminium and Steel have a similar distribution, with more changes along the cylinder height in the Aluminium but more module gradient in the Steel but with a similar distribution. However, Cooper and its opposition to be deformed by flexion loads create a greater deformation in radial expansion instead of adjust its form to the punches shape. This finally is transformed in a bigger radius deformation in module but with similar width in its deformation areas.
- Finally, it is not possible what material is better than other, it purely depends on the purpose that this material is going to be used and the deformations it has to support. Sometimes it I required than materials support this loads without suffering any deformation or on the other side, sometimes it is preferred to cause small deformations in the material in order to make known that something is not working. A good example of this last possibility is working with 4D printed materials, but it is another topic to talk about. What material is used also depends on the ease to be changed and its accessibility.

All these aspects have to be considered before having the decision of what material use in each specific case. If on the contrary, what is required is just to analyse what consequences are going to be in some small parts due to vibration efforts, here there is a very good example to intuit what can happens on them.

ANALYSIS OF THE RESULTS – DYNAMIC MODEL (MODAL ANALYSIS)

Apart from the static analysis done, the main body of this project is to focus on the modal analysis to the parts involved on this test. Firstly the small cylinder is going to be analysed and secondly, the modal analysis of the sonotrode (the most important analysis of this part due to the fact that, in reality, the vibration is provided to this part) will be carried on. This analysis will be carried out in order to know the frequencies that should be avoided to not to have problems with resonance and, actually in our case, to find an explanation to this rotation movement associated to the cylinder

Going directly to the analysis and following the APPENDIX 2 - GENERAL PROCEDURE TO CREATE SIMULATIONS IN ANSYS steps to create the figure and its proper modal analysis, first part to be presented is the small cylinder (this is also done first because of its simple shape and to be more confident with this new program).

CYLINDER

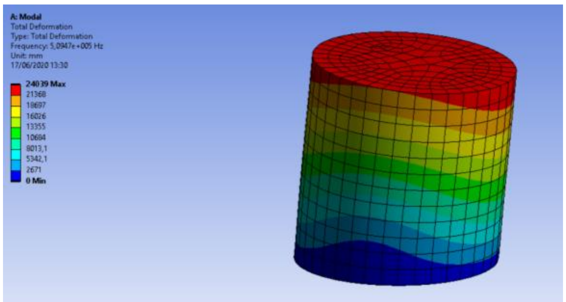
This cylinder is going to be tested in one of the default steel materials given by the program. First of all, the synchronous frequencies are presented below:

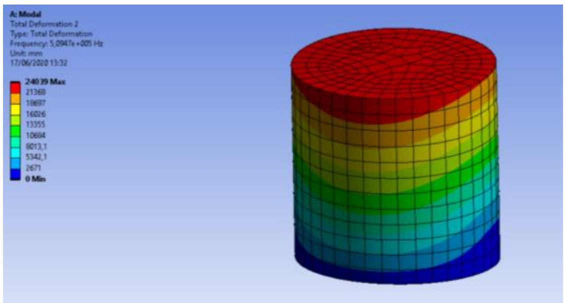
MODE	FREQUENCY [Hz]
1	$5{,}0947 \times 10^5$
2	$5{,}0947 \times 10^5$
3	$7{,}9437 \times 10^5$
4	$1{,}2943 \times 10^6$
5	$1{,}4572 \times 10^6$
6	$1{,}4572 \times 10^6$

Figure 18. Cylinder synchronous frequencies

Due to be a small part, the synchronous frequencies are quite high. Later we will be able to compare them with the frequencies on the two sonotrodes tested. First and second, fifth and last, have the same frequencies, so their states can be considered the same.

Cylinder simulations results are as follow (it is shown the frequency of the different trial made and the colour bands region for each deformation, with the corresponding value for each colour labels on the left):





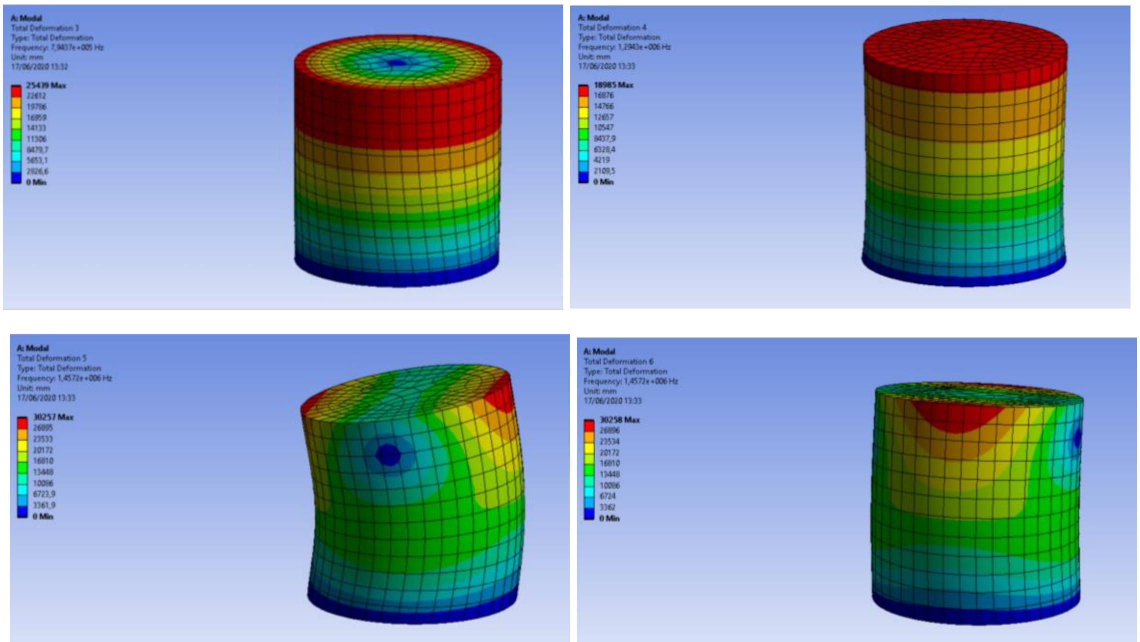


Figure 19. Cylinder deformation graphs

First and second images are the same frequency representation but watched from two different points of view. This fact is repeated again with figures five and six. Results are not going to be commented deeply because this figure is not as interesting as the sonotrode to be studied. Anyway, it is important to realise that frequencies are very high due to its small size.

SIMPLE SONOTRODE

Starting from the first and simplest sonotrode that is going to be tested, the material now is going to be changed to the one that is the real goal to be achieved (the one expressed in the. This sonotrode is used to acquire more knowledge on the program and especially about the geometric formulation. Later, the real sonotrode will be more precisely analysed.

Below it is shown a figure with the dimensions of this sonotrode:

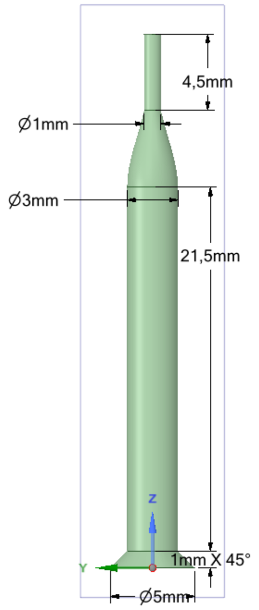


Figure 20. Simple sonotrode dimensions

The material properties required to have the most similar scenario to the target we are trying to achieve are:

Properties	
Appearance	
Color	ARGB: 255, 143, 175, 143
Style	By Layer, By Style
Tessellation Quality Level	5
Material	
Material Name	50026 (50HS)
Fluid	**False**
Density	**0,00778 g/mm^3**
Ultimate Strength (Pa)	**1,6070E+14 Pa**
Elastic Modulus (Pa)	**2,0250E+11 Pa**
Shear Modulus (Pa)	**7,8488E+14 Pa**
Poisson's Ratio	**0,29**
Thermal Conductivity (W/m	
Specific Heat (J/kg-deg C)	

Figure 21. Sonotrode material properties

And now, results of the simulations to find the synchronisation phases on Ansys are represented below the table with the synchronous frequencies presented by Ansys to this shape:

MODE	FREQUENCY [Hz]
1	3189,5
2	3231,9
3	17300
4	17416
5	28646
6	28917

Figure 22. Simple sonotrode synchronous frequencies

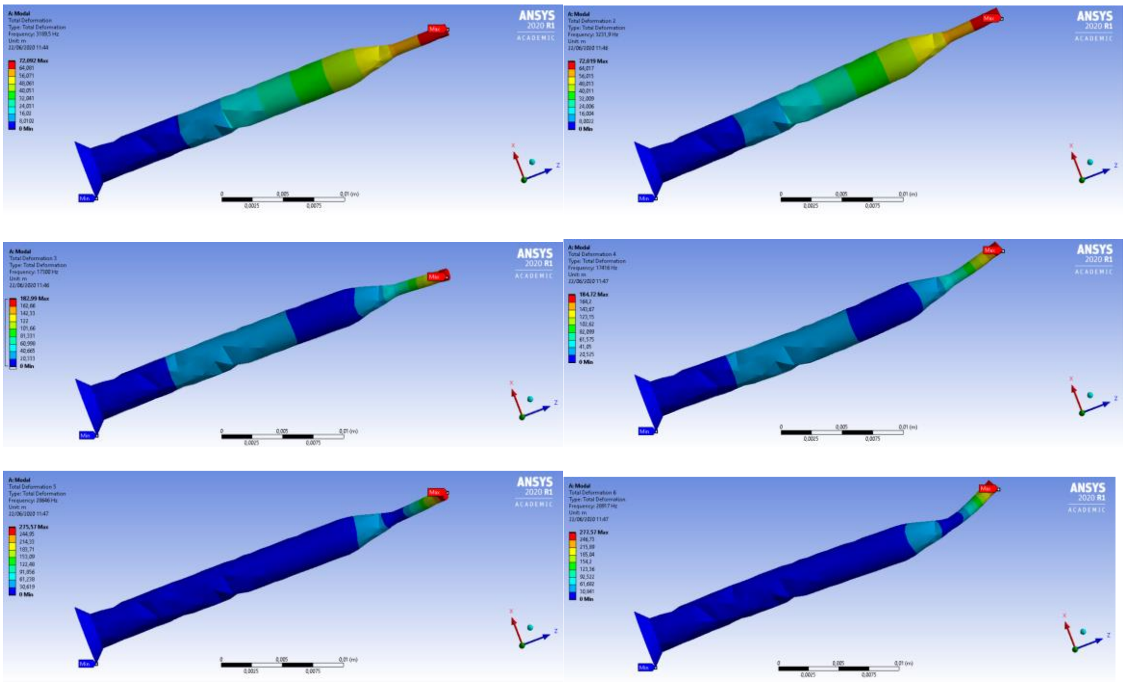


Figure 23. Simple sonotrode deformation graphs

As it is easy to observe, they can be grouped in three main groups with big difference between them but just few Hz of differences within. This is surely due to the symmetry of the shape. Now the most complex sonotrode is going to be tested in a similar way. As it was shown on this sonotrode, the results surely will be quite similar respecting shape factors.

COMPLEX SONOTRODE

Finally, the target sonotrode shape is tested and simulated on Ansys program. The dimensions of this part (the closest to the desired dimensions with the geometric program restrictions and my lack of large knowledge about it) are as follows:

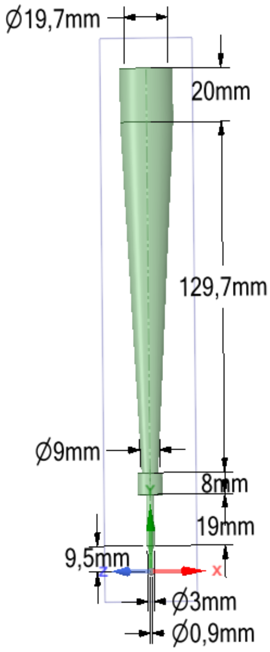


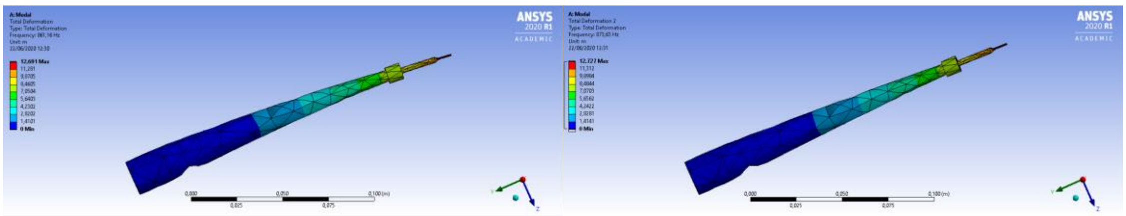
Figure 24. Complex sonotrode dimensions

With this sonotrode shape and the material associated previously defined, the synchronization frequencies are:

MODE	FREQUENCY [Hz]
1	861,16
2	873,63
3	2407,3
4	2441,8
5	45220,8
6	4577,4

Figure 25. Complex sonotrode synchronous frequencies.

As we can observe, the same way as it happened before with the simpler sonotrode, the frequencies are joined by pairs. Each 2 frequencies, there is a big difference in the synchronous frequency value. On this case, due to the higher differences in the cylinder diameters which compound the whole part, the differences in frequencies are also higher.



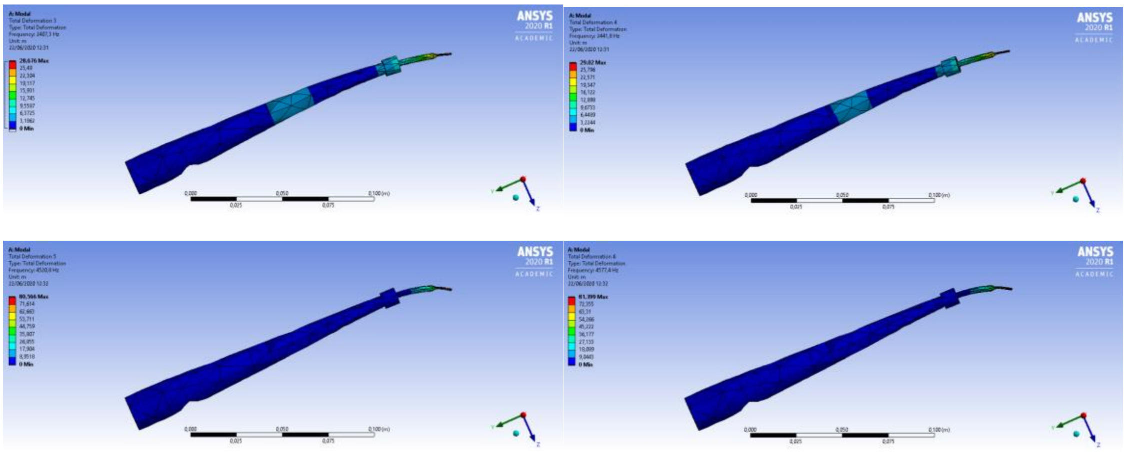


Figure 26. Complex sonotrode deformation graph

As it is possible to be seen, the same way as it happened before with the simpler example sonotrode, the more the frequency grows, the less area has the greatest deformation gradient. This fact is very important because in the last frequency experimented in the simulation, the highest deformation happens in the thinner part of the sonotrode, this can cause the break of the last part, the one that transmit the vibration to the small part tested.

Due to the importance of understanding the size of this really small part and the even smaller size of the cylinder tested, it is shown below the real scale size and shape of this sonotrode:

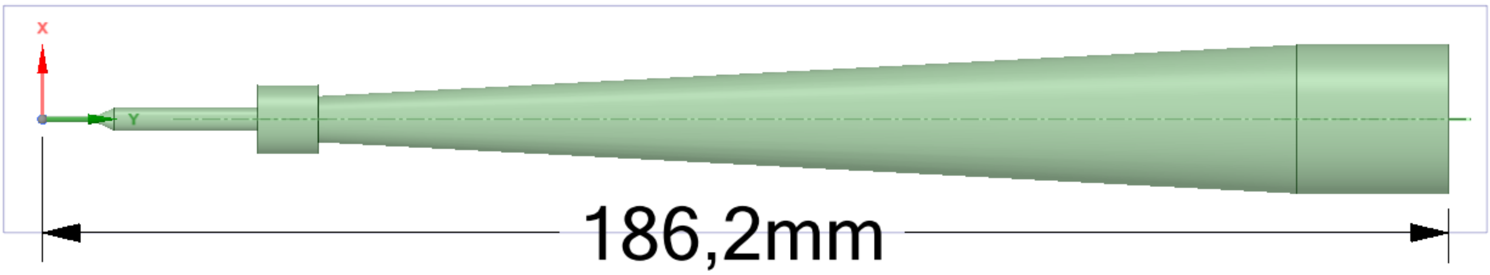


Figure 27. Real sonotrode size

In this way we can appreciate the importance of details in these microforming movements (the thinner part is the one that causes this deformation on the cylinder).

MODAL ANALYSIS CONCLUSIONS

Despite the fact that only three simple analysis has been developed, some conclusions can be drawn:

- The smaller dimensions the part has (mainly talking about the length), the higher frequencies will be needed to find the resonance/synchronization ones. This is due to the ease that compact object has to be deformed by external forces. This is possible to be summed up as, the more equal the different dimensions in a figure are, the more difficult to destroy this object by vibration will be.
- On the other hand, as we could observe with the sonotrodes. They are much longer than the small cylinder and their synchronization frequencies are much lower.

- It is also important to realise that in these sonotrodes with their particular shapes, as we are growing in frequency, their deformation gradient happens in a more reduced range. This is dangerous because the pressure increases exponentially in the thinner part of the figure.
- Finally just telling as an information commentary that the frequencies are high compared to normal vibrations in actual life, but they are several powers under expected, as it was presented in *Presz, Wojciech* documents. The lowest frequency achieved in the real sonotrode that can cause synchronization is under 900 Hz. This makes much easier to reach the synchronous frequency than expected, as it was just commented. This means that with a lower frequency, it is possible to have an explanation to this strange phenomena of the rotating part.

POSSIBLE ROTATION EXPLANATION

The target of this report is to find an explanation of the rotation movement about why a normal vibration movement in a determinate frequency can cause rotation in a stationary object if this vibration is applied in a direction parallel to the height of this part (as the cylinder previously presented).

Now, once Modal analysis of the sonotrode has been presented, some valuable conclusions can be achieved (these conclusions are just based on the last of the three simulations lately presented). As it is possible to observe on the last picture of all englobed on the Figure 26, the maximum deformation (in the top front, where we are interest on) is equal to 81.4. Thinking on it as a percentage of the total displacement associated to the whole sonotrode, we can observe that the end of this part is reaching 81.4 of this displacement to each direction, so in a plane, the total movement is double of that. According to this assumptions, it is possible to establish the movement out of the exact centre of the target part as follows:

$$Total\ movement = 81.4\%\ of\ 16\ \mu m = 13.024\ \mu m \xrightarrow{2\ sides\ of\ the\ plane} 26.048\ \mu m$$

The end of the sonotrode moves on the upper surface of the cylinder part with a diameter of 26.048 μm. This means a diameter almost 40,000 times smaller (exactly 38,390) than the whole diameter of the cylinder can make it start rotating with help of the normal vibration caused by the synchronous frequency and inertia forces. This fact, next to the orbital microforming, explains the final form that normal cylindrical shapes can achieve due to these two phenomena applied at the same time.

Apart from this, there exists another important fact which is the jumps or lack of contact between the sonotrode and the cylinder. This fact also helps to this rotational movement. It looks like to be in a chaotic way, but it is very periodic, so the shape, suffer the load in non-centric position as it has been above mentioned and joined with the non-continuous touch, possibly created the beginning of this rotational movement.

CONCLUSIONS

The conclusions will be divided into three main sections. In the first one, the problem dealt with in this work will be commented on, followed by aspects related to the programmes used and finally, the methodology followed to deal with this work.

- In the first place, it seems to me, particularly, a very interesting subject at the same time as complex the one treated in this work (together with my colleague Ignacio Sánchez Francisco, in charge of treating the orbital part) and that requires much more time and knowledge than we have. It is perfectly possible to dedicate an entire course only to the analysis of this topic and the different variants that exist in it. From my point of view, dealing with this topic that seems so remote as it is the forming by deformation due to vibration plus orbital efforts is essential to understand and above all to foresee possible failures in small parts around engines and bodies that work at high revolutions and consequently create different frequencies of such vibration and orbital, this has been widely discussed in the introduction. As I have said, there are many things to take into account when drawing clear conclusions, for example, in my particular case working with vibration I had to choose (mainly because of time issues, I did not have time to make many variations) between several aspects such as frequency of the vibration, amplitude of the vibration, advance of the movement, associate it to the position of the punch or on the contrary do it to the speed or to the load exercised, radius of curvature of the two punches used, initial dimensions of the shape to model, final state that I wanted to obtain of this shape, and a long etcetera.
- Changing the subject, and now focusing on Marc Mentat, I have a bittersweet taste. The program is very powerful when it comes to trials of this kind. You can do an endless number of trials with many variables collected. In the same way, very variable results can also be collected in terms of load, position of nodes or points, different total or partial pressures, etc. I am sure I could have treated more data and possibly of greater interest. This is why the feeling is not entirely good as far as this program is concerned. The student version, which is the one we have used, has certain real limitations and not only when it comes to limiting the total number of elements in the mesh, but sometimes it fails despite having all the parameters within a totally logical order. In spite of this version, which surely has not been one of the main problems, the one that has been is the ignorance that we had and that we still have in front of this powerful program. Having to start from scratch and on our own totally with such a laborious program has made us lose a great deal of time in tests that have been modified later or that I implemented at the moment of truth I did not compile or run when we wanted to do the final work for each simulation. By this I mean that the data collected in this work is just a small part, the salvageable part, of what we worked on in this program, which is several hours a day.

  On the other hand, talking about Ansys Program, the second program used to find the modal analysis simulations on the different parts, it is important to comment the ease on its use in good and bad sides. It is very easy to implement the desired form (once the base knowledge is acquired), but it is not easy to find complex results due to this reason. The program is quite simple for everything, for creating the shape and for the analysis. Time dedicated to this program has been clearly lower than the dedicated to Marc Mentat, but results are closer to what we were looking for. Keeping working on this program maybe result could be even closer to the reality and make a more complete analysis of the results obtained.

- On this last point, as is the methodology used when dealing with this problem, it must be made clear that the circumstances are very particular. Having done everything through simulations due to the pandemic that is affecting everyone makes the work too theoretical and there is no evidence in the practical part, as is understandable. As I commented in the first of these three points, this is a very extensive work and many variants must be taken into account, many of which I have not even mentioned such as temperature, humidity and so many other aspects that affect the behaviour of materials under these loads. Treating some of these tests in a practical way, in a laboratory, would have given us a more realistic idea of what we were looking for and what we were achieving, as it was all very idealistic. As for how the information on the objectives to be pursued was transmitted to us, I do not think it was the best option. Having a weekly meeting was very good because it monitored our progress (even though I would have needed more, especially at the beginning), but I needed a guide to follow from the beginning to the end, stating the objectives of the work and focusing on how they should be treated.

  Originally, this work was going to be joined to the work carried on by Sánchez Francisco, Ignacio related with the orbital and both together find the common solution we are looking for by mixing vibration and orbital movements, but due to the lack of time, it has been impossible.

Finally, I would just like to say that I find this topic more than interesting and useful in my work and that I will continue developing it after this issue because, as I said, it is of my personal interest. The programmes I used surely I also use it in other occasions because, as I said, I think it is very complete in many topics, not only in vibration and material deformation studies. Taking all this into account, I would like to express my satisfaction for the accomplishment of this work and for the conclusions gathered.

APPENDIX 1 - GENERAL PROCEDURE TO CREATE SIMULATIONS IN MARC MENTAT

GENERAL PROCEDURE

The problem treated on this report is associated with the study of the deformations in a small cylinder shape due to vibrations loads applied by one, two or all necessary presses. This problem is analysed because of it use to happen very frequently in current mechanical parts cause of the vibration some electric or combustion engines create on their working process, but this fact has already been analysed in the introduction. Now, the procedure followed to create the different simulations is going to be developed and later the distinctive features will be analysed for each particular state.

First of all, the points that are going to determine the part and press bounders have to be defined. In every moment we are going to work in axisymmetric conditions, so it is important to define just half of this parts and later associate them to the axisymmetric axis, which has to be also created on this point. This points and different types of lines have to be created with following command:

Geometry & Mesh > Basic manipulation > Geometry & Mesh

Line can be created between 2 points by putting the number of these points or if points are visible, by clicking directly on them.

If the design is very simple, it can be rapidly created by putting a grid before and directly clicking on the points of this grid. To put the grid, the instruction is very simple:

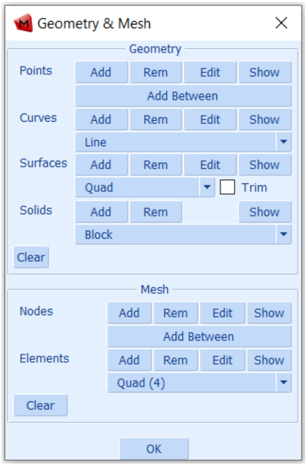


Figure 28. Table of Geometry and Mesh MM

Geometry & Mesh > Coordinate system > (click on the box) *Grid* -> ✓

It is possible to edit the grid field by clicking *edit* under the grid mark.

Once the part (usually half cylinder) has been created and the press/presses it is possible to create the mesh. Cylinder will always have an individual mesh and press can also have mesh or not depending on the study level we want to achieve and if we need a real press with depth or a simple line can simulate this press in the 2D simulation. To create these meshes it is obligatory to create the curve division attending to our particular interest and straightaway associate the mesh to these divisions. This process has to be done as follows:

Geometry & Mesh > Pre – Automesh > Curve divisions + Choose target length or number of divisions and select lines associated to this characteristics.

Geometry & Mesh > Automesh > Planar + Quad mesh and select the body to be meshed.

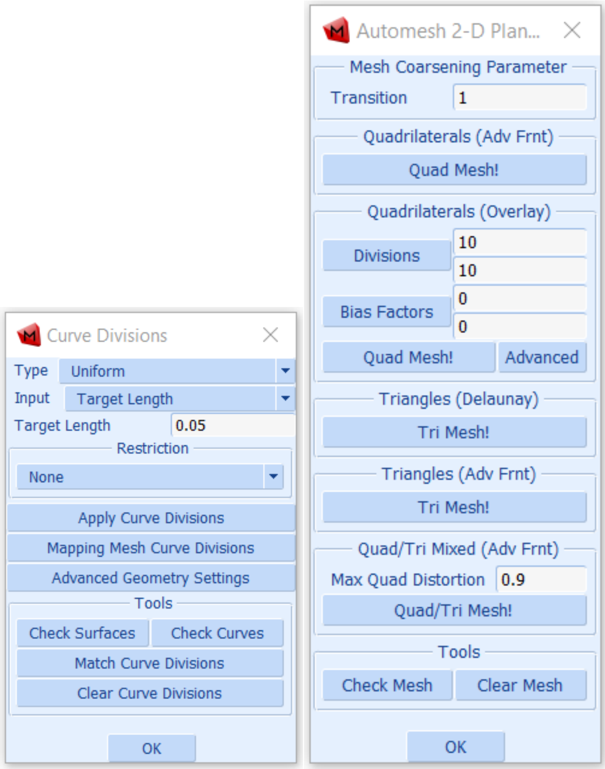


Figure 29. Tables of curve division and automesh MM

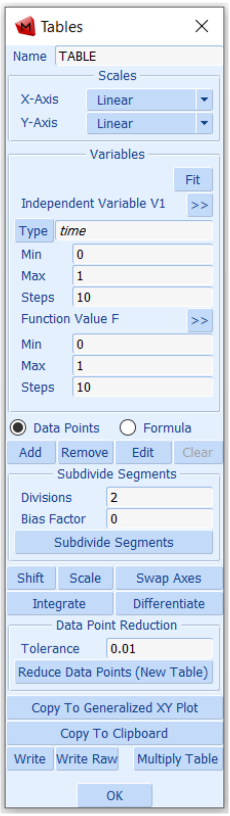


Figure 30. Tables about movement tables

Following with the natural order of the Marc tabs, the next one is Tables & Coord. Syst.

On this step the table to associate the movement of the press or directly to the cylinder has to be defined. It will be later explained but we will associate this table to the velocity variable because this makes the movement more similar to what we are looking for.

Tables & Coord. Syst. > Tables > New > 1 Independent Variable

Type will be usually set as *time*, but it is important to know that this time is not as we think about normal time due to the lack of mass in Static simulations, this is just a variable to make program run.

The more *steps* in each axis, the more precision we can achieve.

Both *Data Points* and *Formula* are going to be needed in order to have the perfect movement. Mainly *Data Points* to bring closer and distance the press from the part at the beginning and at the end of the simulation and *Formula* to define more precisely the main movement we want the press to do.

Continuing with the following tab, here we are going to select the structural model that is going to be used. As it was said before, it is going to be the axisymmetric one because this one fits perfectly with the study we are going to do.

Geometric Properties > New (structural) > Axisymmetric > Solid + Select all elements

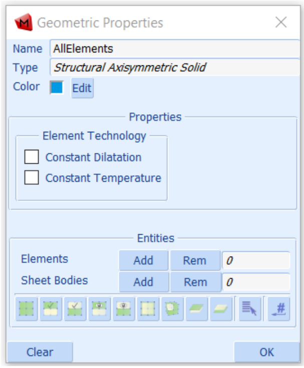


Figure 31. Geometric properties MM

Next step is to define the materials used. This is selected by introducing all the properties for the different materials we want to use independently.

2 main materials are going to be used usually, aluminium and steel.

Material Properties > Material Properties > New > Finite Stiffness Region > Standard + Select Mass Density, Young's Modulus, Poisson's Ratio and Plasticity with Yield stress.

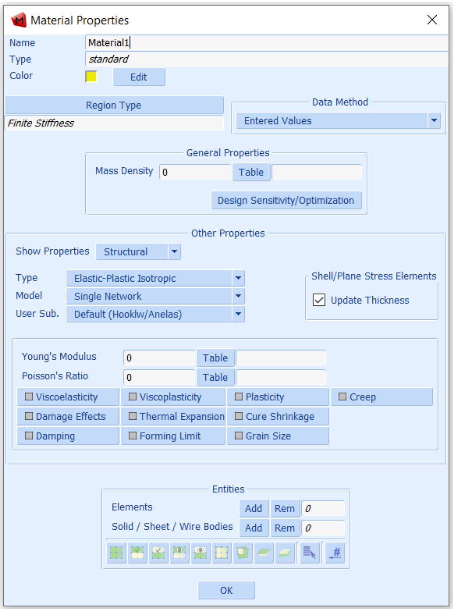


Figure 32. Material Properties MM

Values selected for each material are discussed in other point of this report, but it is relevant to mention that it is very important to pay attention to the units use by this problem.

Mass/Loads have to be set in Kilograms, Time in seconds and Distance in millimetres.

Next step is define the contacts among the different bodies and each body properties. For this fact, first of all it is important to have a clear idea about the body which is going to be deformed and in our case it is clearly the cylinder in every simulation.

For the cylinder:

Contact > Contact Bodies > New > Meshed deformable + Select all elements related to the cylinder.

Contact > Contact Bodies > New > Meshed rigid + Select press if it is in 3D defined. If there are more than one press, create one contact body for each of them.

Contact > Contact Bodies > New > Geometric + Select press if it is line defined.

Contact > Contact Bodies > New > Symmetry + Select symmetry lines, at least the axis of the cylinder.

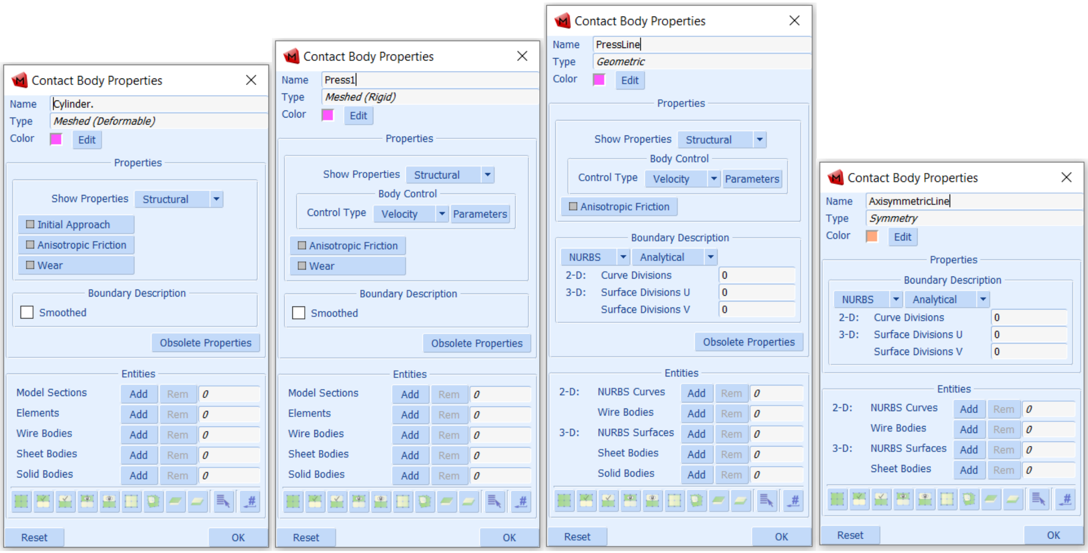


Figure 33. Contact body properties MM

On this tables the loads, velocities and positions can be associated. It can only be done in those bodies which have *Body control > Control Type* in their table. On this new option, direct loads can be associated to these bodies or previous created movement tables can be chosen:

Figure 34. Contact body control MM

Now bodies are totally defined, but not still iterations among them.

We obtain a better response of the system by adding a velocity instead of a load or a position definition. We estimate this velocity by derivation of the load function selected.

Contact > Contact Tables > New + define all iterations. It is easier to define all of them as touching and once this has been selected, change them by a different way. When it is selected a kind of iteration, a new section appears in the index description: *Contact Iterations*. Here is where iterations are easier to be changed and friction coefficient can also be selected.

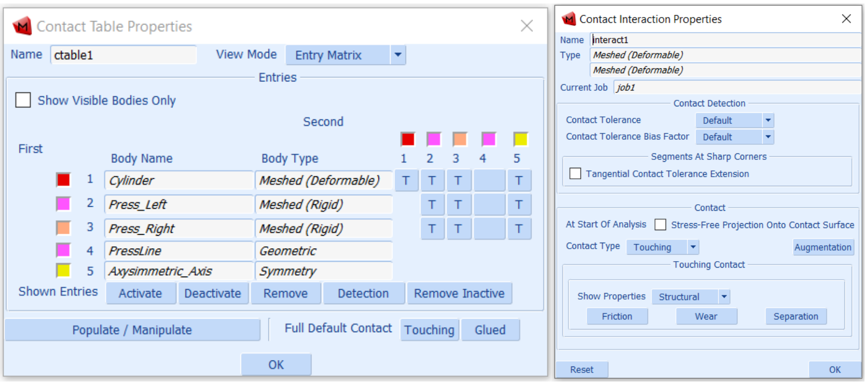


Figure 35. Contact table and interactions properties MM

Finally, to be able to compute results, it is necessary to create the job, for this it is necessary firstly to create a loadcase.

It is also quite important to mention that the friction coefficient that is going to be implemented in all the contacts (without taking into account the materials that are working) is going to be set as 0.2.

Loadcases > Loadcases > New > Static (usually)

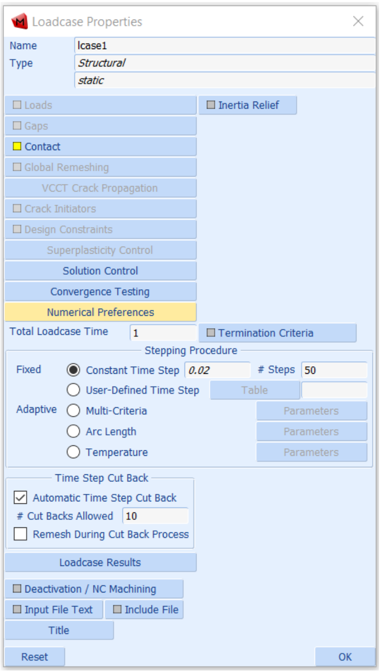


Figure 36. Loadcase properties MM

On this table it is important to put the proper Loadcase time, Contact time steps and total number of steps in order to have the wished simulation.

After this, finally, the job can be created.

Jobs > Jobs > New > Structural (usually). Steps to follow on this point are very mechanical.

Select the desired loadcase (there use to be only one created in each document and if it is needed to create another one, sometimes it is easier to copy the document and change it in this new one).

Select Analysis Options and the type of stress that is going to be treated on this simulation (small or large).

Chose the job results to be analysed and presented from the program.

Contact Control > Initial Control > Contact table + Select the table

Choose *Axisymmetric* as *Analysis Dimension* in Job Properties table.

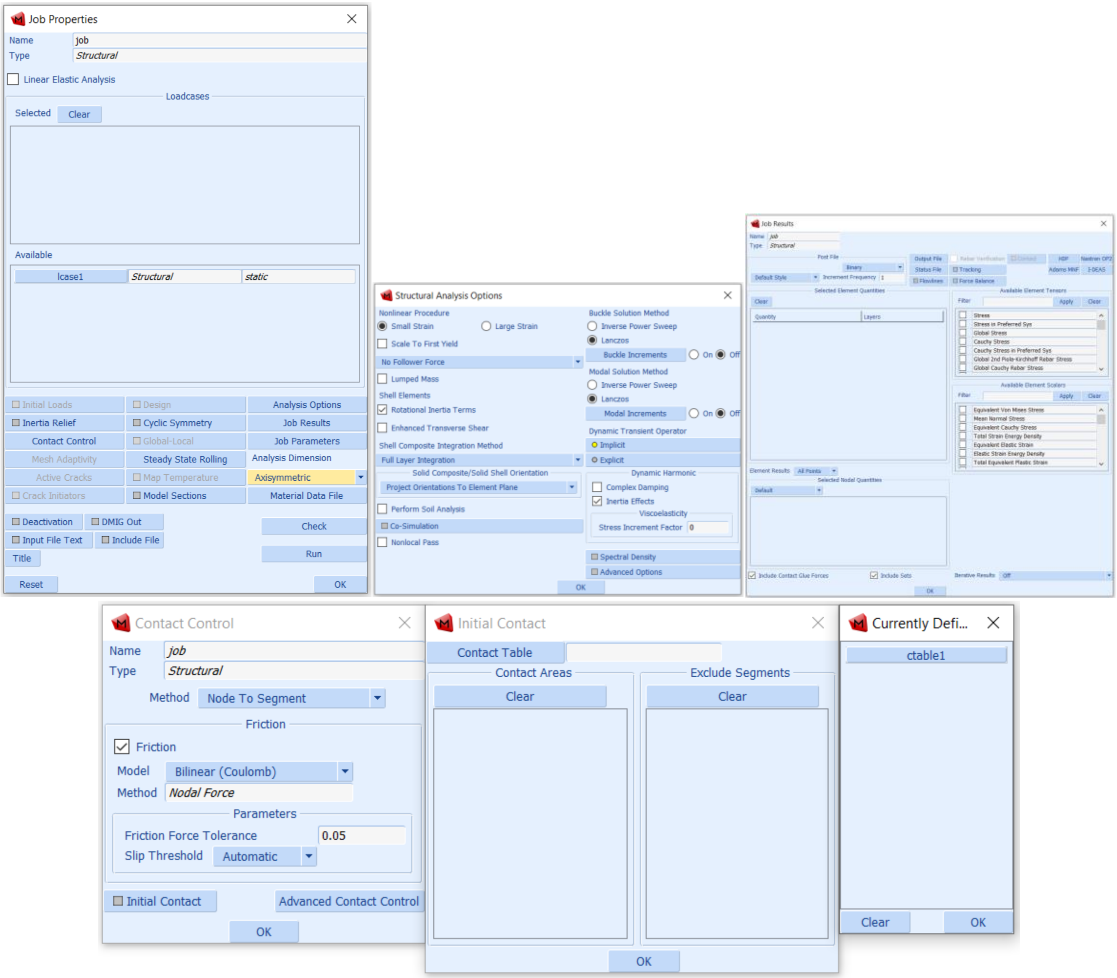


Figure 37. Jobs changes from default values MM

It is also important to mark the asymmetry in the *Jobs > Element Types > Element Types*

On target, *Elements* are leaved as default and *Solid* is selected

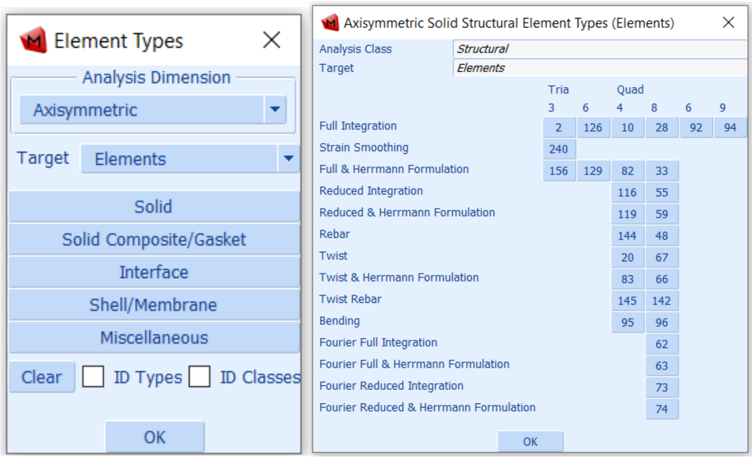


Figure 38. Element types and Structural element types chosen MM

Model to be used is selected (number 10 for example) and all elements are implemented with this model.

By clicking *Check*, program tells if any error or warning has happened while the analysis verification has been made. Click *Run*.

Last table appears:

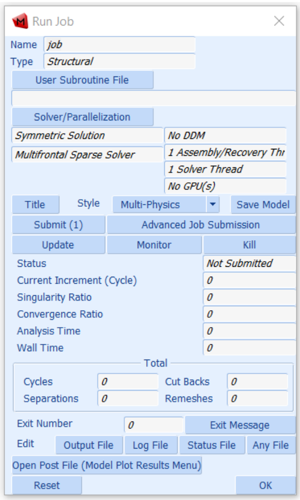


Figure 39. Run job table MM

Following steps (in this Run job table) are:

Save model -> Submit (+ wait till the job is totally submitted) *-> Open Post File (Model Plot Results Menu).* Results are directly presented and visual analysis is selected with the following table:

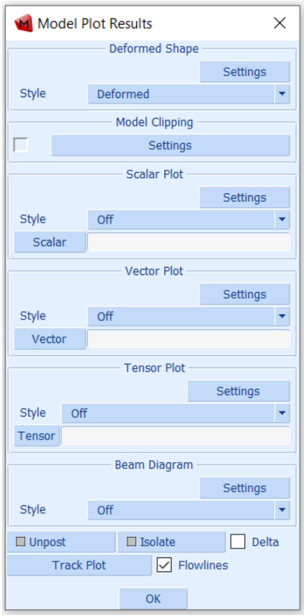


Figure 40. Modal plot results MM

On this table above we can select different results to be shown on the simulation process among the ones selected previously in the Job results table (Figure 37 - A) and some other given by default.

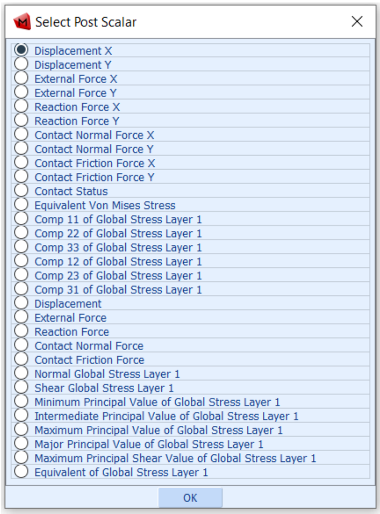


Figure 41. Post scalar to show in results table.

EXTEND THE 2D PORTION ALONG THE AXIS IN VISUALIZATION.

In order to have a visualization of the process, it is useful to Represent a larger visualization of the part built. This means to expand this part on a wider range then the shell usually represented on the usual 1 degree representation.

Geometry & Mesh > Operations > Expand

Rotation Angles (Degrees) along the required axis (in this case X Axis).

For example 5 degrees along this axis.

Additionally, choose the number of *Repetitions* for this degrees representation. For example 36.

This case we will have 5 degrees part multiplied by 36 times, what means, 180 degrees part. Half of this part will be represented and it is nearer a 3D simulation.

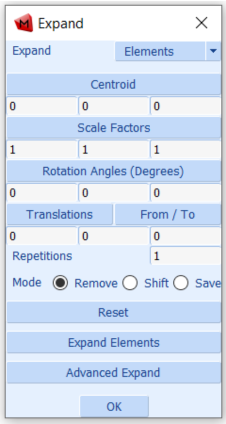


Figure 42. Expand table MM

MATERIAL PROPERTIES AND UNITS

Units used by the Marc Mentat and Ansys programs:

- Length – Millimetres
- Weigh – Kilograms
- Loads – Newtons
- Time – Seconds (actually it is not important due to just working on static mode)
- Angles – Degrees
- Density – Kilograms / Cubic meter

ALUMINIUM MECHANICAL PROPERTIES:

$$Density \rightarrow D = 2700\frac{kg}{m^3}$$

$$Poisson's\ ratio \rightarrow P = 0.32$$

$$Young\ Modulus \rightarrow Y = 70\ GPa = 7 * 10^{10}\ Pa = 7 * 10^4\ MPa$$

$$Tensile\ Yield\ Strength \rightarrow TS = 183\ MPa = 1.83 * 10^8\ Pa$$

COPPER MECHANICAL PROPERTIES:

$$Density \rightarrow D = 8960\frac{kg}{m^3}$$

$$Poisson's\ ratio \rightarrow P = 0.4$$

$$Young\ Modulus \rightarrow Y = 119\ GPa = 1.19 * 10^{11}\ Pa = 1.19 * 10^5\ MPa$$

$$Tensile\ Yield\ Strength \rightarrow TS = 33.3\ MPa = 3.33 * 10^7\ Pa$$

STAINLESS STEEL MECHANICAL PROPERTIES:

$$Density \ \rightarrow D = 7850 \frac{kg}{m^3}$$

$$Poisson's\ ratio \ \rightarrow P = 0.29$$

$$Young\ Modulus \ \rightarrow Y = 200\ GPa \ = \ 2 * 10^{11}\ Pa = \ 2 * 10^{5}\ MPa$$

$$Tensile\ Yield\ Strength \ \rightarrow TS = 215\ MPa \ = \ 2.15 * 10^{8}\ Pa$$

DIFFERENT MATERIALS SIMULATION RESULTS.

Starting point for every simulation is the one that follows:

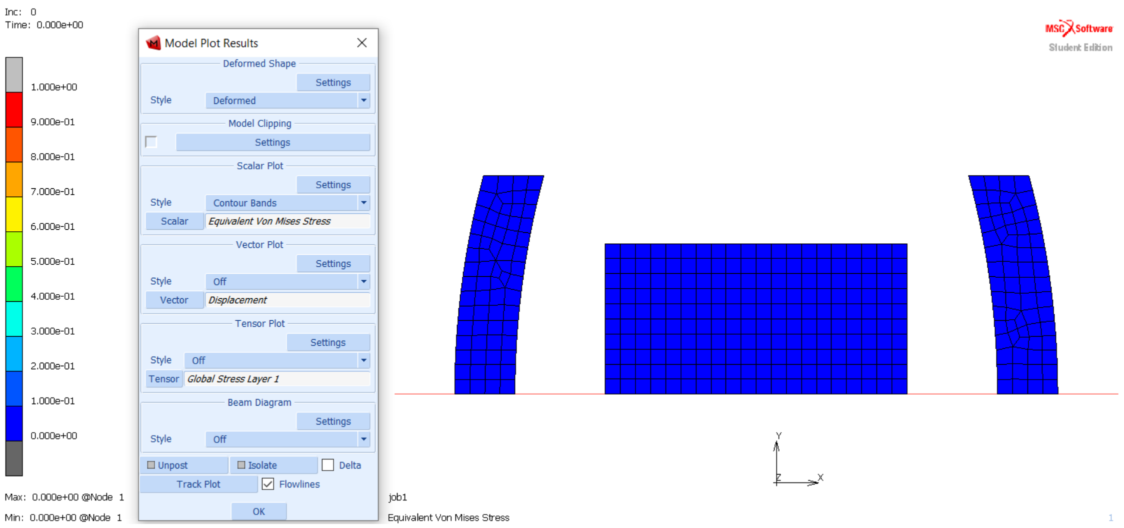


Figure 43. Simulation results starting point

Advance of the vibration will start at the beginning of the shape form and it will cause a reduction of the cylinder starting from 1 millimetre along the all cylinder to finally achieve an external height of 0.768 approximated in the piece external length and half of this reduction in the middle line, reaching a minimum length of 0.884 approximated.

In Figure 44, it is possible to appreciate the difference between starting and final states.

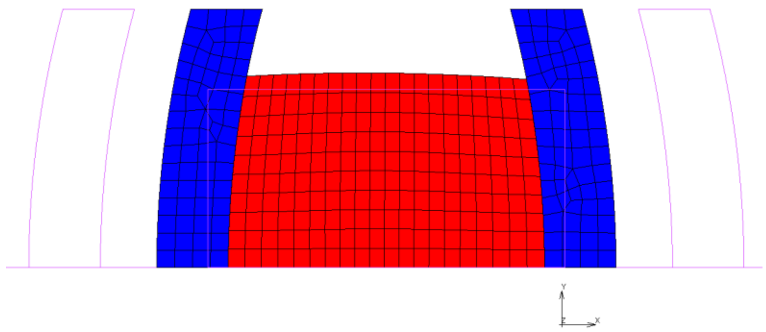


Figure 44. Difference between starting and ending point in simulations

ALUMINIUM SHAPE

Starting the analysis with the aluminium shape, most deformable material of the three chosen it is expected to find a lower stress load needed to deform this material. This is tested by looking at the load when the compression is executed. This happens in the 2 left picture below, first of them showing the process just in the middle of the time range and the second row images the last compression and the final state. In both compression cases, the ones where the shape is all in red (what means that all of this part is subjected to the shape load) have a von Mises stress equals to $1.87*10^2$ MPa, what is higher than the $1.83*10^3$ MPa of Yield stress of the material.

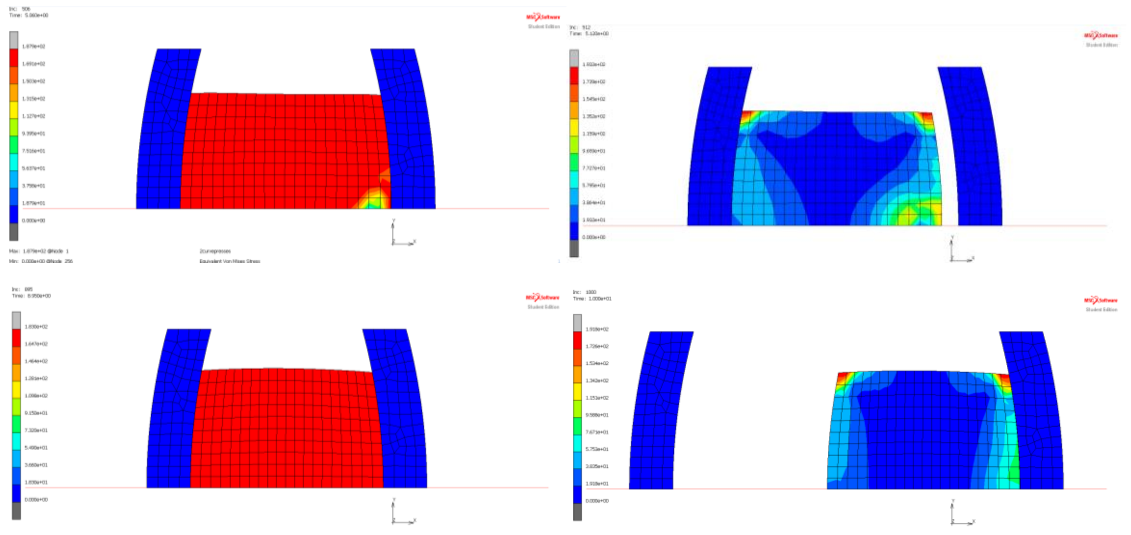


Figure 45. Aluminium stress distribution

The residual stress is maximum in the corners of the simulation, what represents the external edge of the bottom surfaces (external circumference of the circular faces). Moving forward in the analysis it is also easy to observe the stress accumulated in for bottom surfaces and few low stress also in the external curved surface of the shape. The internal part of this shape has not suffered huge changes from the beginning of the movement and at the end of the simulation, it is shown its neutral state referred to residual loads.

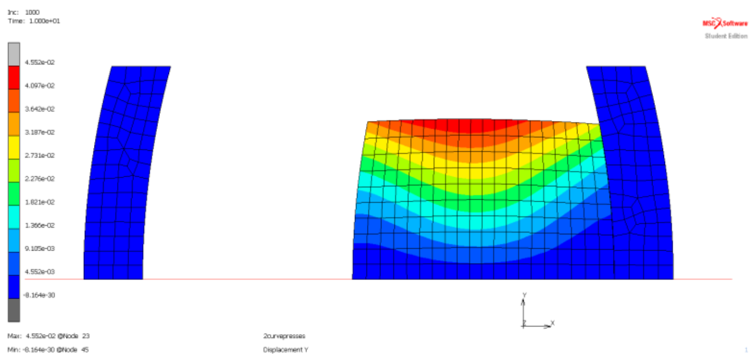


Figure 46. Aluminium radius displacements/expansion

About the displacements along the Y axis, one axis associated to the radius direction how displacements are maximus in the middle of the cylinder and they are a bit lower in portions nearer to the tops of the shape. The negative value of the scale has not to be considered because in a symmetric load application it is not possible to have value that overpass the symmetric axis.

COOPER SHAPE

Continuing with the analysis, material is changed from Aluminium into Cooper to analyse the results and the differences with previous example.

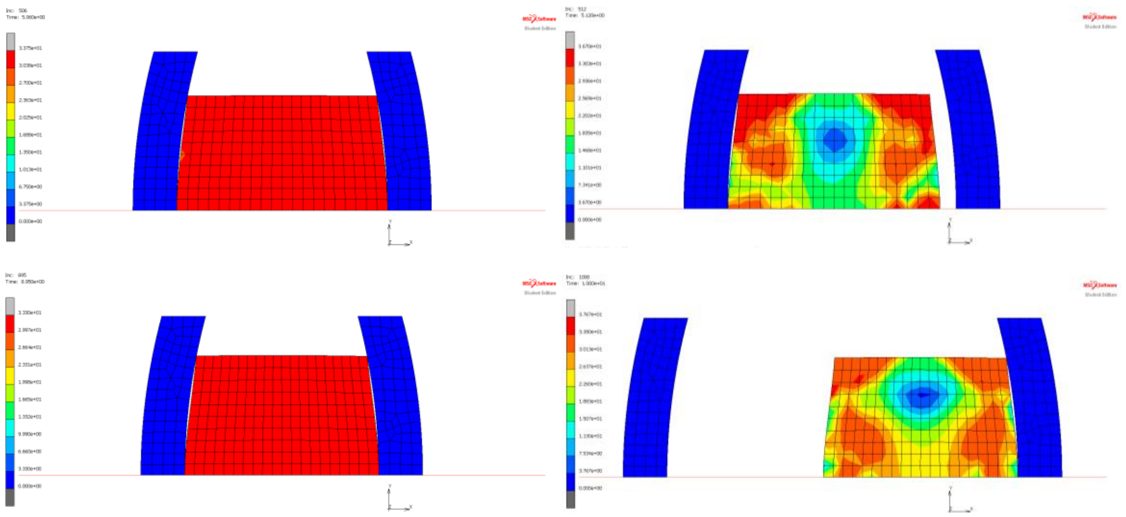


Figure 47. Cooper stress distribution

It is very remarkable the new shape of the cylinder when it is compressed. Due to the fact that it can afford much easier the compression stress than the flexion ones associated in this case to the effort created by the shape of the curved punches, the reaction of the cylinder to this compression vibration is to grow in radius magnitude and not to adjust to the punches shape. The stress distribution when the load is not active and pressing the shape is very different to the one of the previous experiment. Much more residual loads can be observed in the final state of the cylinder.

Maximum stress at compression states (again the two on the left) have a total value of $3.37*10^1$ MPa more than the Yield stress of this material ($3.33*10^1$)

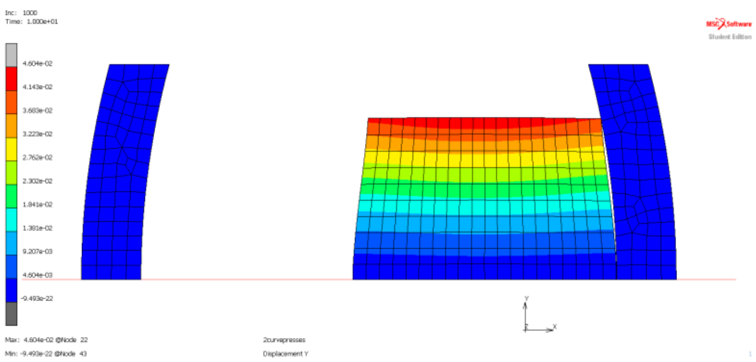


Figure 48. Cooper radius displacements/expansion

Displacements in this case have a quite different distribution. Areas have a straighter look. This is for the same reason commented above, the strength of the material. This strength is higher than in Aluminium and that is why material can easily support compression efforts but not flexible ones. Cause of this, it is also observable that the higher displacement is bigger than in the previous material simulation.

STAINLESS STEEL SHAPE

Finally, the last material is going to be analysed, Stainless steel. This material has the highest Yield stress of all of the tested ones, so surely more stress will be needed in order to cause the required deformation.

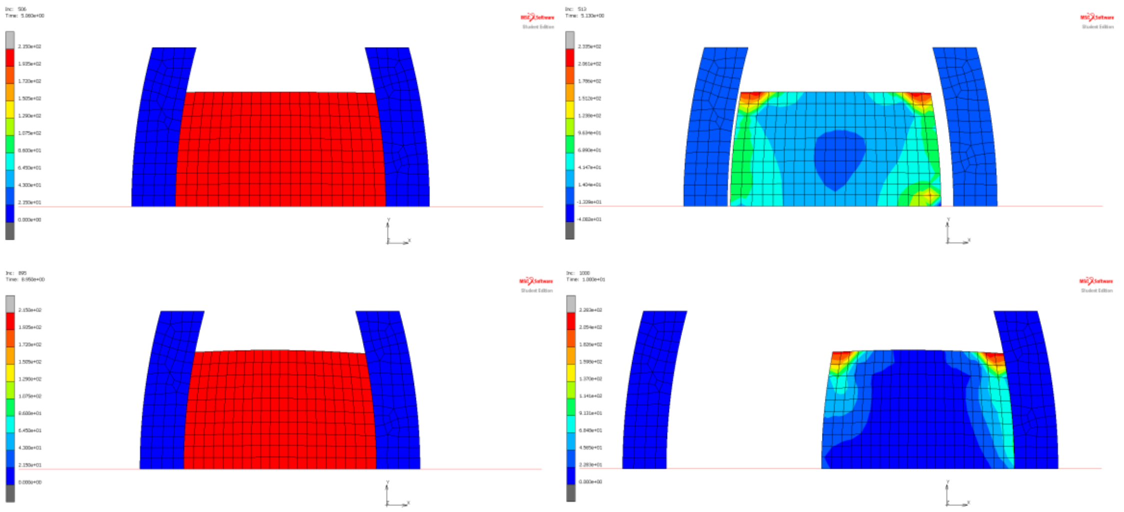


Figure 49. Stainless steel stress distribution

As it was forecasted, the stress when the punches are in the high peak of their wave is around 2.15*10^2 MPa, the Yield Stress of this material. In rest states (when the load is not active, so the two pictures of the right) the stress distribution in the cylinder is quite similar than in the Aluminium case, saving the magnitude of this stress. Final part has the same residual stresses with maximum values at the corners and some displacement along the lateral an bottom surfaces with a middle of this part with a non-stressed state.

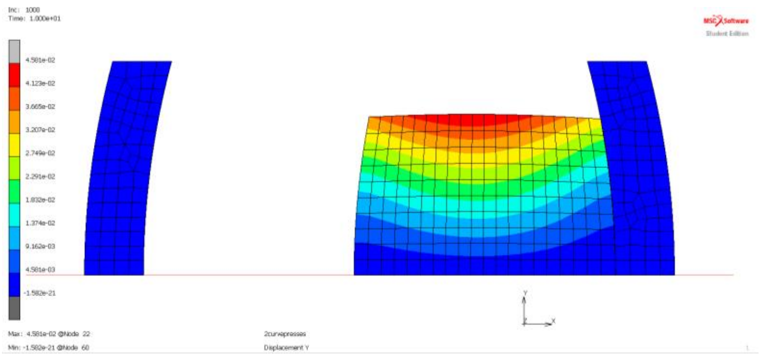


Figure 50. Stainless steel radius displacements/expansion

Talking about the displacements of the elements in the extension of the radius direction caused by the compression of the punches are similar to the happened in the Aluminium case but with not so drastic changes, so we can place this result between the two previous results. Anyway, its maximum value is higher than in Aluminium case. This is caused to the proper characteristics of the aluminium to be compressed and reduce its shape by compression efforts and loads.

(Engineering Stress Intuition, 2018; Software MSC, 2013)

APPENDIX 2 - GENERAL PROCEDURE TO CREATE SIMULATIONS IN ANSYS

The steps that has to be followed in order to create a figure following description dimensions and make its modal analysis to know the synchronous frequencies that it has are going to be described on this point. The program used because of the difficulties found in Marc Mentat is going to be Ansys.

Figure 51. Ansys front page

First of all, the analysis that we want to visualise at the end has to be chosen. In this case, modal analysis with dynamic properties → MODAL.

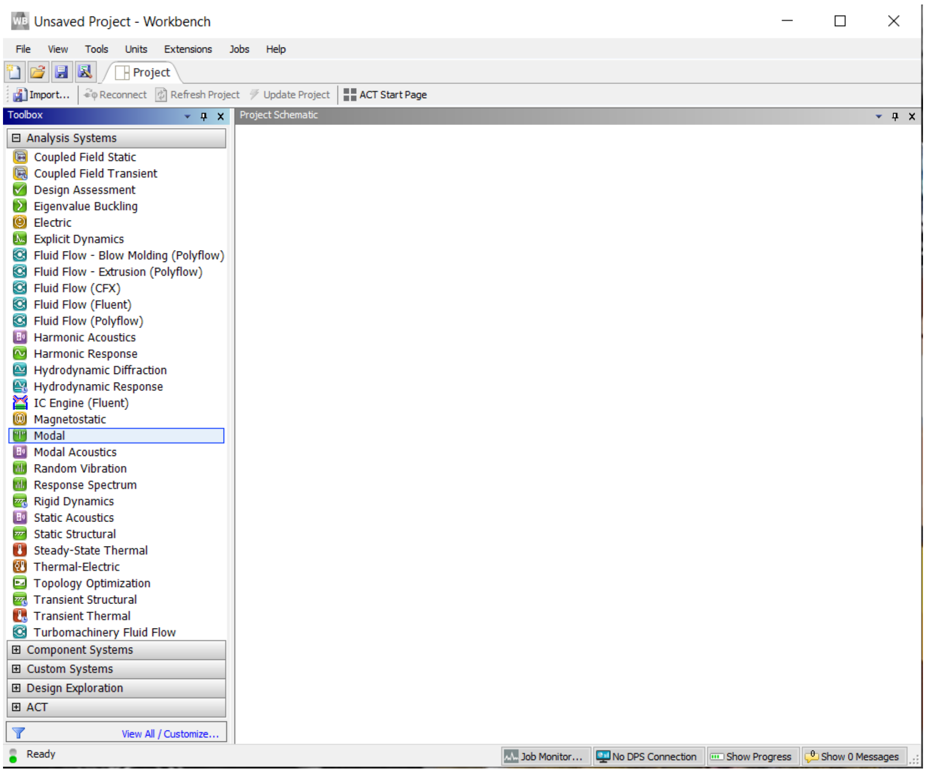
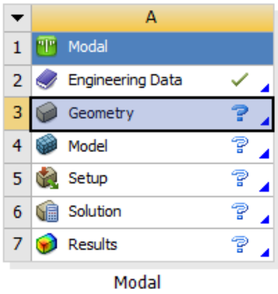


Figure 52. Starting page Ansys

Next dialogue box will appear and by choosing geometry, there will be the possibility of creating the part shape.



This fact makes another (and different) application of the Ansys program opens and allows to create the physical part by overlapping solids, specifically, creating surfaces, joining them to create the solid and finally, joining the separated solids to create a pool. Tab where create the solid with the sketch visible and the final part are shown below. In this case the solid represented is the simple sonotrode in order to have a clear idea of the steps followed.

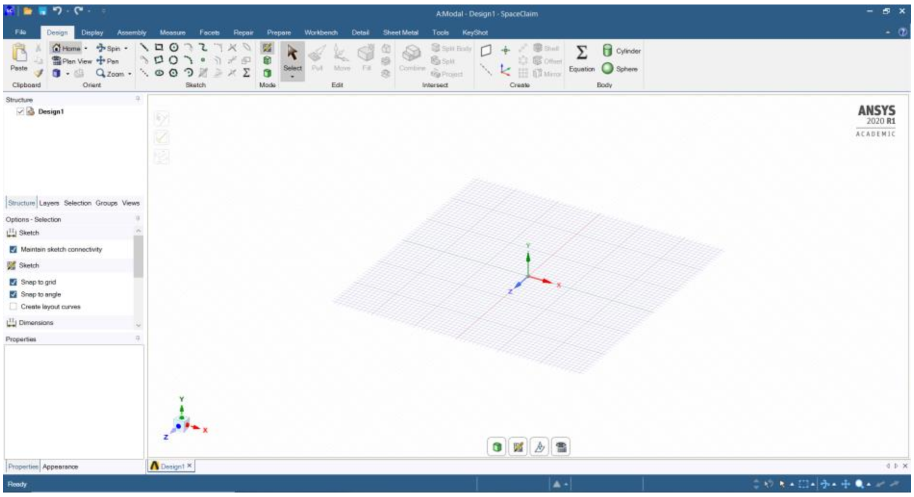


Figure 53. SpaceClaim Ansys

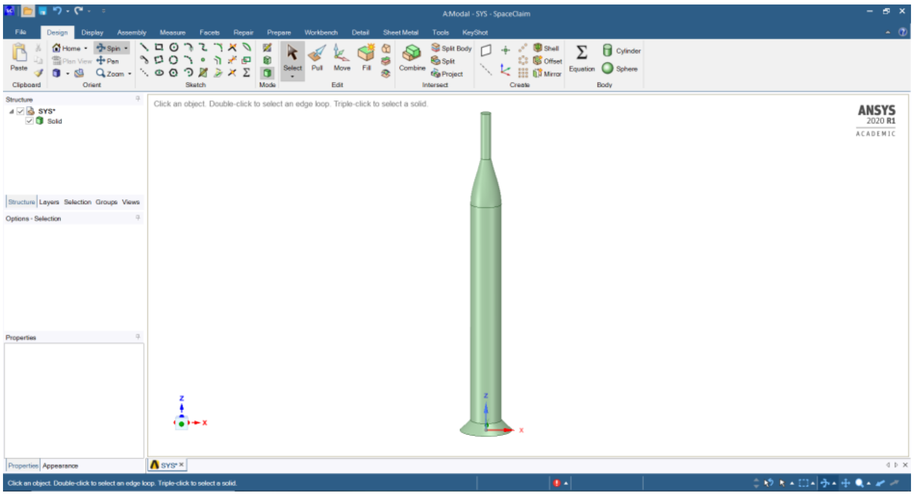


Figure 54. Figure created in SpaceClaim in Ansys

On the following part of the upper menu, next to Workbench, there is a tab telling Detail. There it is the possibility of adding the dimensions of the created solid, not to change the dimensions, just notice the dimensions previously chosen.

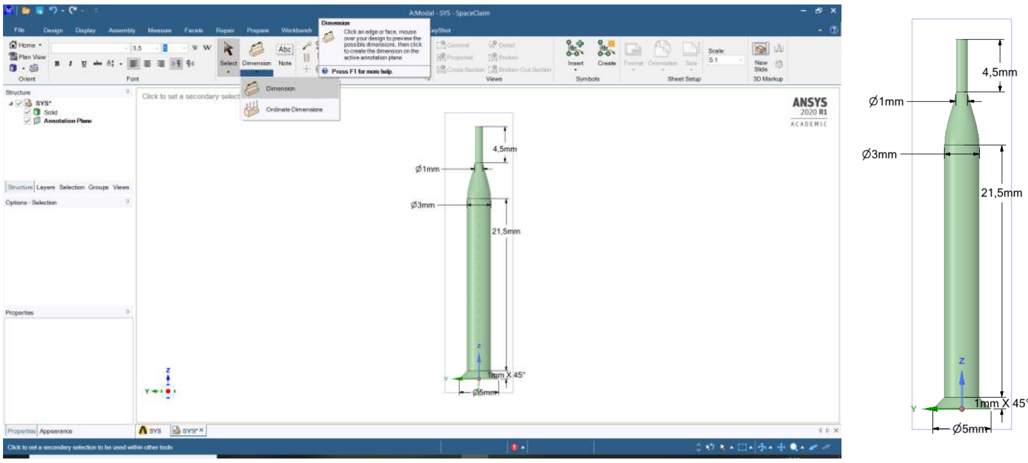


Figure 55. Figure dimensions (Ansys)

Part of creating the solid shape is finished. If any specific material has to be implemented, the steps to follow are quite simple. First of all it is needed to press the solid we want to put the material (in this case we just have one solid) and directly by clicking on the properties, it is possible to choose one of the materials given by default or type specific mechanical properties.

To test this created shape, the next step is to open again (actually it has not to be closed) the workbench WB application, and without closing the spaceclaim SC (the building application), open the model application as it is shown below.

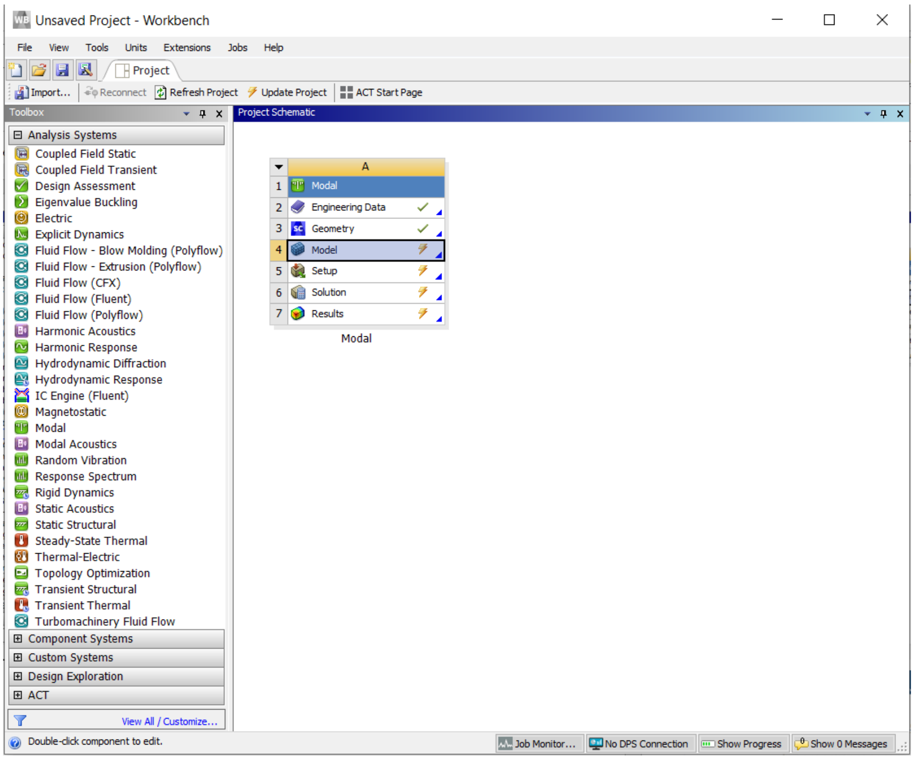


Figure 56. Results application Ansys

Solid previously created will directly appear in the screen and now it is time to define the parameters that we want to have controlled, in this it will not be a very difficult analysis.

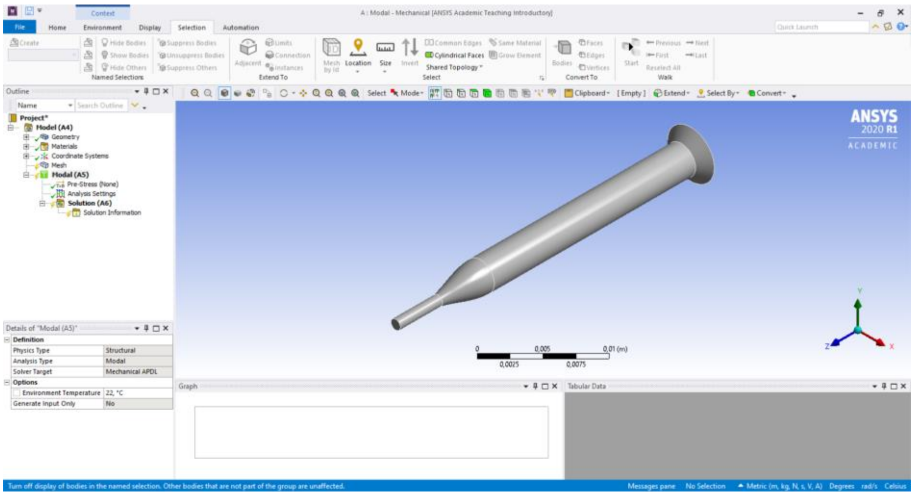


Figure 57. Modal Ansys

Firstly, the fixed base has to be defined.

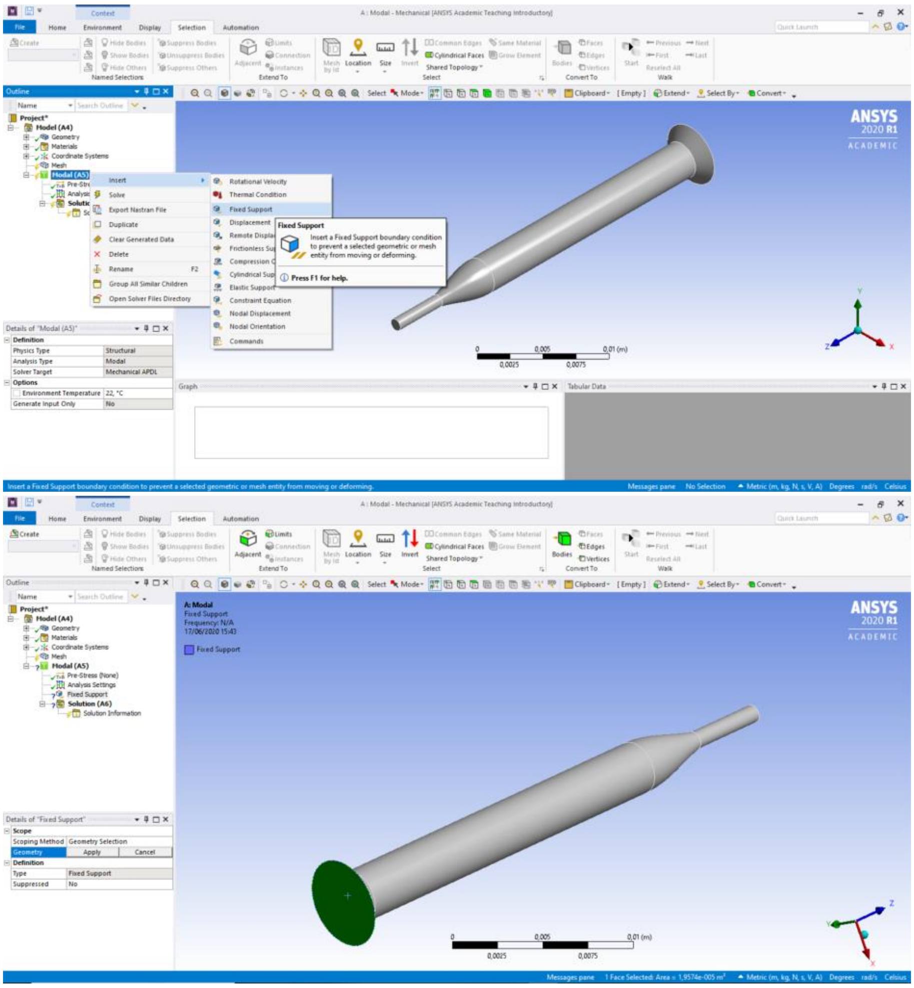


Figure 58. Fix base Ansys

Left click on the surface + Apply selection (2 arrows icon). Following blue figure shows that this point (and consequently the surface) has been fixed in support = displacement.

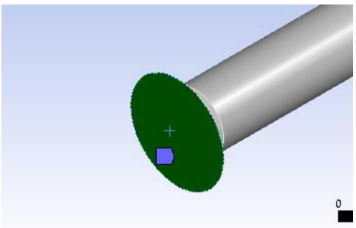


Figure 59. Fix base confirmation Ansys

To know the different phase with resonance, solve bottom has to be pressed:

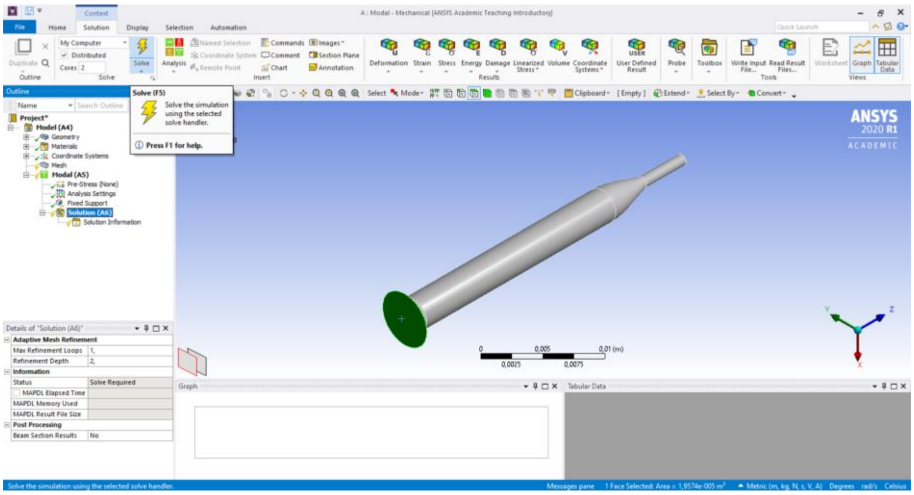


Figure 60. Solve Ansys model

Automatic mesh is created:

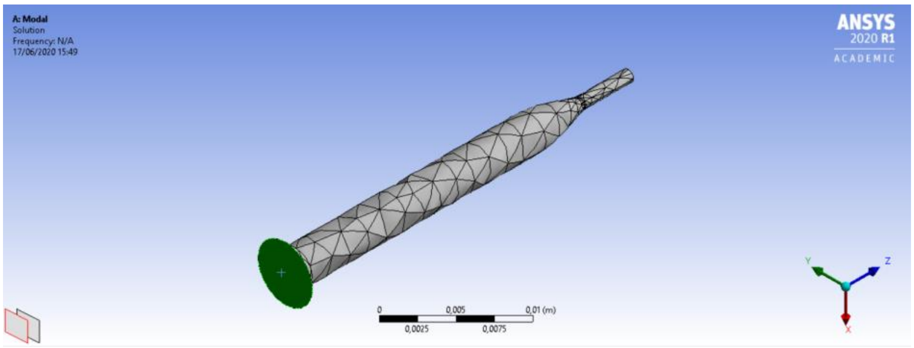


Figure 61. Meshed figure Ansys

And results with their respective frequencies are shown below the figure:

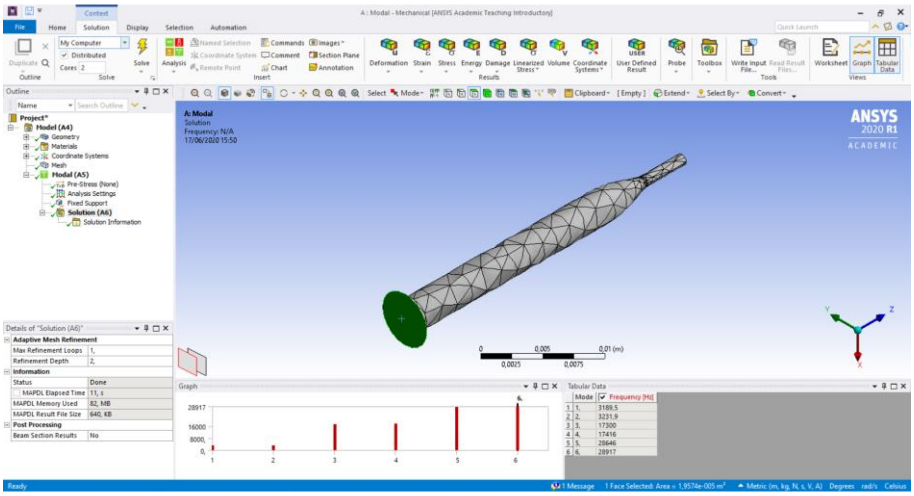


Figure 62. Final representation pre-results Ansys

By the following action we are going to be able to visualise them (after selecting the whole range):

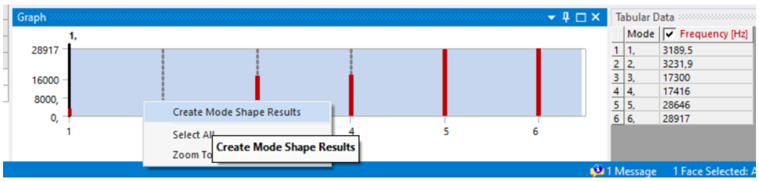


Figure 63. Computed results and representation guide Ansys

In the left index will appear the different cases and clicking on them the visual result, but before being able to visualise them, the evaluation process has to be followed (shown in second below figure).

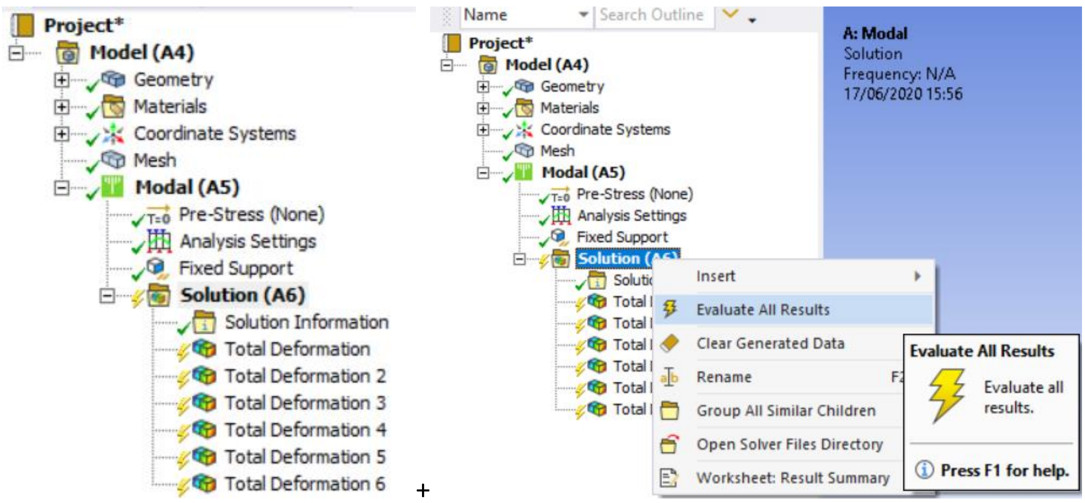


Figure 64. Results evaluation Ansys

(*Ansys Modal Analysis - tutorial (old version of the program)*, n.d.)

LIST OF FIGURES

REFERENCES

Ansys Modal Analysis - tutorial (old version of the program). (n.d.). Retrieved June 15, 2020, from https://www.youtube.com/watch?v=1qnaRzWuAZI&t=507s

Engineering Stress Intuition. (2018). *Door Seat Nonlinear FE Analysis using mentat*. https://www.youtube.com/watch?v=qVDUTr-gu1E&t=610s

Geiger, M., Kleiner, M., Eckstein, R., Tiesler, N., & Engel, U. (2001). Microforming. *CIRP Annals - Manufacturing Technology*, *50*(2), 445–462. https://doi.org/10.1016/S0007-8506(07)62991-6

Mechanical Resonance. (n.d.). Retrieved May 31, 2020, from https://en.wikipedia.org/wiki/Mechanical_resonance

Modal Analysis. (n.d.). Retrieved May 31, 2020, from https://en.wikipedia.org/wiki/Modal_analysis

Modal Analysis using FEM. (n.d.). Retrieved May 31, 2020, from https://en.wikipedia.org/wiki/Modal_analysis_using_FEM

Presz, W. (2019). Planetary riveting of electric bimetallic micro-contacts. *Procedia Manufacturing*, *27*, 83–90. https://doi.org/10.1016/j.promfg.2018.12.048

Presz, W. (2020). Influence of experimental setup parameters on ultrasonic assisted micro-upsetting. *Archives of Metallurgy and Materials*, *65*(1), 423–731. https://doi.org/10.24425/amm.2020.131745

Presz, Wojciech. (2018). Ultrasonic orbital microforming—a new possibility in the forming of microparts. *Metals*, *8*(11). https://doi.org/10.3390/met8110889

Presz, Wojciech. (2019). Material flow in ultrasonic orbital microforming. *Metals*, *9*(4). https://doi.org/10.3390/met9040475

Project, I. I. (2020). *Ultrasonic Orbital Microforming (Orbital forming)*.

Software MSC. (2013). *Marc - Simulating Manufacturing Processes with MSC Software solution*. https://www.youtube.com/watch?v=kM_-n6jchQM